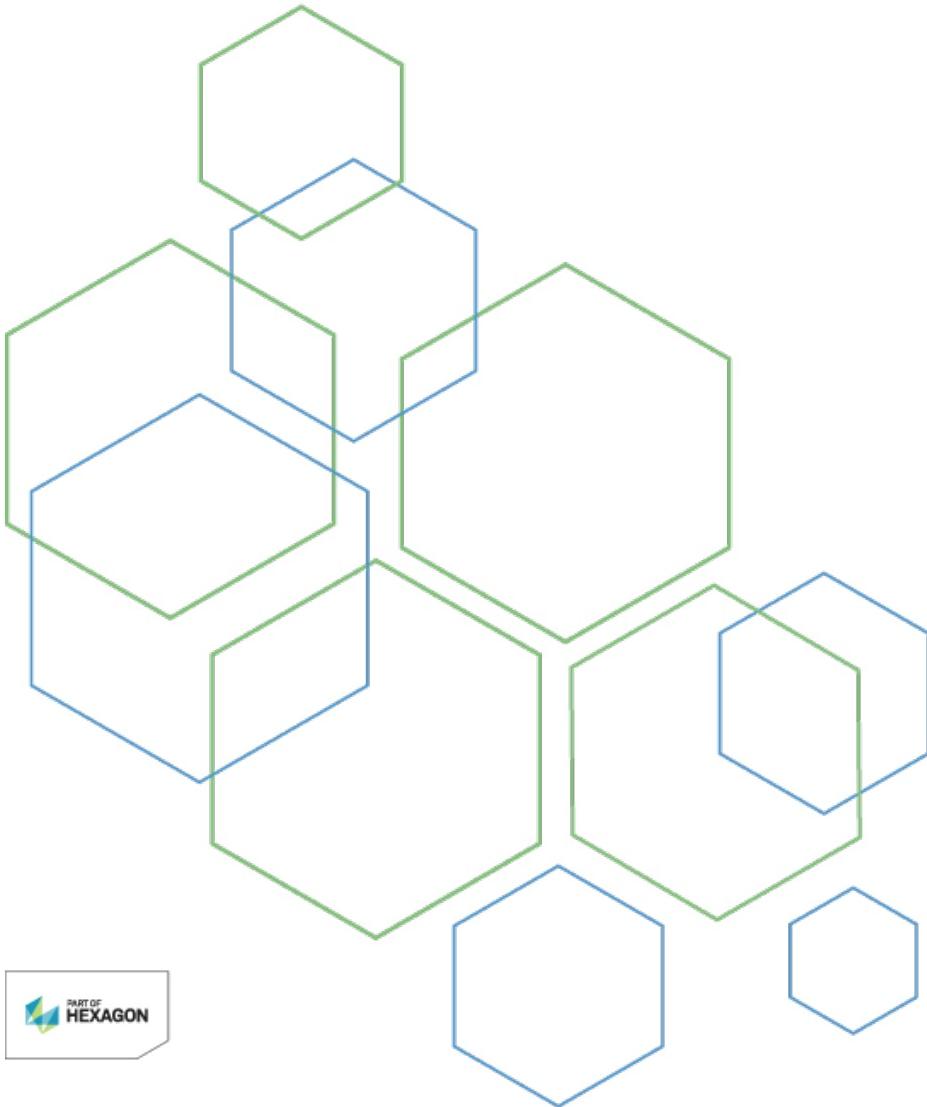


INTERGRAPH®

Smart[®] 3D

Equipment and Furnishings

User's Guide



Version 2016 (11.0)
November 2016

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Preface

This document is a user's guide for the Equipment and Furnishings functionality of Intergraph SmartTM 3D and provides command reference information and procedural instructions.

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (<https://smartsupport.intergraph.com>).

What's New in Equipment and Furnishings

The following changes have been made to the Equipment and Furnishings task.

Version 2016 (11.0)

- A new property, **SuppressGraphics**, specifies whether or not to display pipe nozzle graphics. For more information, see *Occurrence Tab (Nozzle Properties Dialog Box)* (on page 111). (P2 CP:238958)
- Added a new locate filter, **Construction Graphics**. For more information, see *Editing Properties and Relationships* (on page 146). (P2 CP:271166)
- Added a new option, <**Select Graphically**>, to the **Place Equipment** ribbon. For more information, see *Place Equipment* (on page 18). (P2 CP:271451)
- You can now place an imported shape from a VUE file. For more information, see *Place Imported Shape from File* (on page 125). For information about VUE file limitations, see *Place an imported shape* (on page 126). (P2 CP:280048)
- When you use **Place Imported Shape from File**  to place an imported shape from a .sat, .dgn, or .vue file on the symbol share, Smart 3D remembers the relative path to that file. To save the relative path with imported shapes placed with an earlier version of Smart 3D, you must remodel those shapes. For more information, see *Place Imported Shape from File* (on page 125) and *Share imported shapes between models* (on page 128). (P2 CP:296346)
- You can now change the parent object for multiple shapes at the same time. Previously, you had to change the parent of shapes one by one. (P3 CP:162443)
- In the **Place Equipment** and **Place Designed Equipment** commands, if a parent system is selected when you start the command, the **System** property is automatically updated with the selected system. For more information, see *Occurrence Tab* (see "Occurrence Tab (Equipment Properties Dialog Box)" on page 27) and *Occurrence Tab* (see "Occurrence Tab (Designed Equipment Properties Dialog Box)" on page 44). (P3 CP:251230)

SECTION 1

Equipment and Furnishings

During Model design, you can use the Equipment and Furnishings task to select equipment objects from the Catalog database and position occurrences of these objects in the model using the **Place Equipment** command. The software can also automatically orient the mounting surface of the part during placement to whatever surface you select in the model. After placement, you can precisely position the equipment relative to other parts, structural features, or grid systems within the model by defining additional positioning relationships or by using the **Rotate Equipment** command.

Typically, several pieces of equipment are placed in an approximate position, and then the position of those parts is refined relative to each other and the surrounding structure. After the parts are in their final position, structural foundations are added using the Structure task, and piping connections are made using the Piping task.

The equipment definition in the Catalog consists of a geometric model of the equipment and a set of descriptive properties. The model geometry is sufficiently detailed to serve the purposes of three-dimensional visual recognition and interference detection. The equipment models include detailed definitions of the piping, electrical, and HVAC connections. These connections can be located by the appropriate applications. The software delivers a catalog of basic equipment with the software, including commonly used parts such as pumps and storage tanks, but you can customize this standard catalog or modify the default equipment to match the specific needs of your project.

NOTES

- Nozzles can be shown or hidden independently of parent equipment.
- You can apply styles to nozzles in the same way that you can apply styles to any other object.

Using Visual Basic, Solid Edge or the Equipment and Furnishings task's designed equipment capability to create the graphical 3D representation of the equipment and using the Catalog task to define properties for the parts, you can create any type of equipment that you might need. You can even create parametric equipment models that can be resized after the object is placed in the model. The equipment that you add to your Catalog can be referenced and used on any number of separate designs.

You can start the Equipment and Furnishings task by clicking **Tasks > Equipment and Furnishings**. The Equipment and Furnishings task has the following task-specific commands:

- **Select** - Used to select objects in the model. For more information, see *Editing Properties and Relationships* (on page 146).
- **Place Equipment** - Specifies a piece of equipment from the Equipment folder of the catalog and places an occurrence of it inside the model. For more information, see *Place Equipment* (on page 18).
- **Place Designed Equipment** - Places equipment types that have been defined in the reference data. For more information, see *Place Designed Equipment* (on page 38).

-  **Place Equipment Component** - Specifies an equipment component from the Equipment Components folder of the catalog and places an occurrence of it inside the model. For more information, see *Place Equipment Component* (on page 54).
-  **Place Designed Equipment Component** - Places equipment components that have been defined in the reference data. For more information, see *Place Designed Equipment Component* (on page 68).
-  **Place Shape** - Adds additional shapes or equipment objects to an existing designed equipment type. The icon displayed on the toolbar is the icon of the last shape selected from the palette. For more information, see *Place Shape* (on page 71).
-  **Place Nozzle** - Adds different types of ports to a designed equipment object. For more information, see *Place Nozzle* (on page 106).
-  **Place Imported Shape from File** - Adds geometry to a designed equipment object that was modeled with solid modeling software and saved to a format. For more information, see *Place Imported Shape from File* (on page 125).
-  **Rotate Equipment** - Repositions equipment by rotating along a universal axis defined for the model view as well as an occurrence axis unique to the equipment itself. For more information, see *Rotate Equipment* (on page 131).
-  **Replace Equipment** - Exchanges a selected piece of equipment or equipment component in the model for a different item from the catalog. For more information, see *Replace Equipment* (on page 134).
-  **Place Designed Solid** - Creates customized solids and equipment using shapes that add material to or subtract material from the designed solid. For more information, see *Place Designed Solid* (on page 135).

See Also

Editing Properties and Relationships (on page 146)

SECTION 2

Equipment and Furnishings Workflow

All equipment objects are placed in the model using information defined in the equipment and furnishings reference data. Using the reference data workbook, you can create custom equipment objects. Your first step should be to review, edit, and otherwise customize the delivered equipment and furnishings reference data. For more information, refer to the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command within the software.

After the reference data is customized to suit your needs, consider going to the Systems and Specifications task, or right-clicking in the **Workspace Explorer** and selecting **Create System** to define the systems in your model. You are not absolutely required to create your systems first; however, doing so keeps you from having to edit your equipment objects after placement to assign them to the correct system.

After the equipment reference data and the needed systems are defined, you can begin placing equipment objects in your model.

You can interactively customize equipment from the Catalog by adding primitive geometric shapes and nozzles.

You can also use the Equipment and Furnishings task to build customized equipment. Designed equipment objects can be built entirely out of primitive geometric shapes, or an equipment component part can be designed or placed from the catalog to enhance the designed equipment object.

Equipment components provide a more precise method of building up designed equipment than the technique of adding catalog equipment to designed equipment used by previous versions of the software.

See Also

Equipment and Furnishings Common Tasks (on page 11)

Equipment and Furnishings Common Tasks

The following tasks are used frequently in the Equipment and Furnishings task.

Customize Reference Data

- Create new equipment types by editing the **Equipment.xls** workbook. Review, add, and edit material information. For more information, see the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command within the software.

Create Needed Systems

- Create new equipment systems. For more information, see *Create an Equipment System* (on page 15).

Place Equipment From the Catalog

- Place equipment objects directly from the Catalog. For more information, see *Place Equipment from the Catalog* (on page 22).

Build Customized Equipment Definitions

- Combine basic geometric shapes, ports, and properties to model designed equipment objects. For more information, see *Place Designed Equipment* (on page 41), *Place a Shape* (on page 74), and *Place a Nozzle* (on page 107).
- Import graphics from an SAT file as the geometry for a designed equipment object. For more information, see *Place an Imported Shape* (on page 126).
- Use prismatic shapes to create uniquely customized equipment objects. For more information, see *Creating Customized Shapes* (on page 80).

Set Positioning Relationships

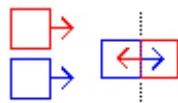
- Creates a relationship between placed equipment and any other surface or reference element that you select. For more information, see *Set Positioning Relationships for Equipment* (on page 23) and *Set Positioning Relationships for Designed Equipment* (on page 42).

Positioning Relationships

The following positioning relationships between equipment and reference elements are available.

Mate

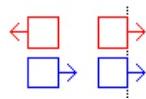
The mate relationship is applied between a surface of the equipment and another equipment surface, structural surface, elevation plane, or grid plane. The surfaces are constrained to be parallel with the indicated offset distance between them. The normal vector pointing out from the surface of one solid points toward the other solid:



The mate relationship can also be applied between the axis of a cylindrical surface and a planar surface. The offset in this situation is applied in the direction of the surface normal. You can use this relationship to orient the equipment's nozzle axis parallel to a wall and offset from the wall a given distance.

Align

The align relationship makes the axes of two cylindrical surfaces collinear or constrains two planar surfaces to be parallel. When planar surfaces are aligned, the surface normals point in the same direction with the indicated offset distance between them:

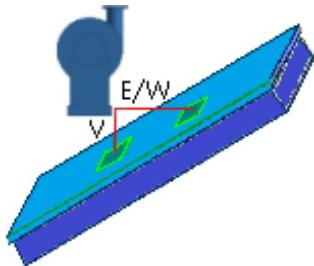


Connect

The connect relationship forces a point on an equipment to be coincident with a point on another equipment, structure, piping part, or arbitrary coordinate in space. The software adds the connect relationship to the relationship list only if the two points connected are both nozzles. Otherwise, the two points are made coincident, but no relationship is saved to the database. Offsets are disabled when establishing a connect relationship.

NOTE Press F3 to turn on or off the **Surface Locate** option. This option makes it easier to create Connect relationships.

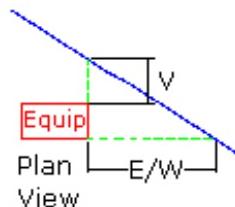
The minimum distance constraint is used to locate a point on an equipment object at a specified distance (horizontal or vertical) from a sloped surface, edge, or point.



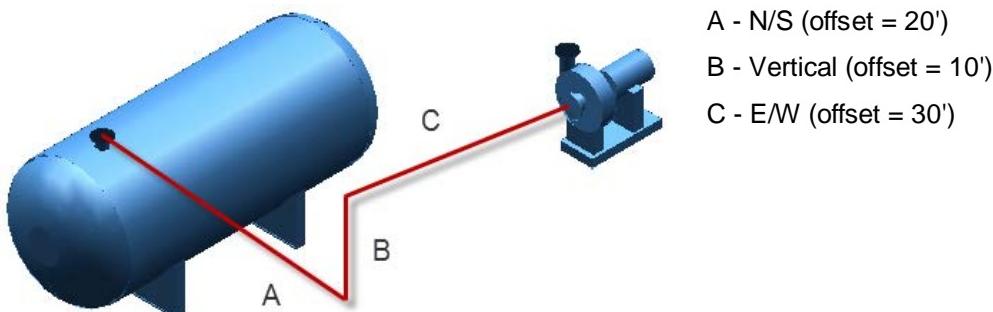
Three constraint options are available: **E/W Distance** (horizontal along the East/West axis), **N/S Distance** (horizontal along the North/South axis), and **Vertical Distance** (vertical along Up/Down axis).

NOTE The referenced axes are in the active coordinate system.

Choose a control point on the equipment that you are placing or editing and a reference surface, edge, or point. Select any control point on the equipment to create the first point of the relationship. The sloped surface can be at any orientation relative to the equipment.

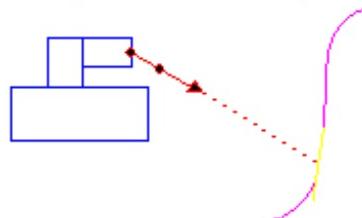


As illustrated in the following example, the three minimum distance relationships can be used collectively to define separate relationships to fully constrain equipment objects.



Mate to Tangent Plane

Using the mate to tangent plane relationship, you can create a tangent plane at the intersection of a user-defined vector and a selected surface. You establish the relationship by first selecting a reference on an equipment object and then selecting two points in the model to define a vector and identify a surface (in the model). The vector is used to locate an intersection point in the model. A plane tangent to the surface (at the intersection point) is



created and used as the mating surface.

Both the vector and the surface are associative inputs; if the referenced surface is modified, the point and the ensuing tangent plane are re-computed.

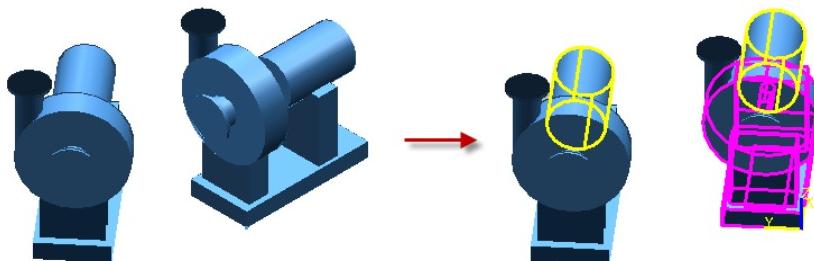
Parallel

Parallel relationships can be set between the following objects:

- Edge to planar surface
- Surface to surface
- Edge to edge
- Edge to axis (implied axis of cylindrical objects)
- Axis (implied axis of cylindrical objects) to axis (implied axis of cylindrical objects)

In terms of behavior, the parallel relationship is similar to align. However, while the align relationship rotates and moves an equipment object, the parallel relationship only rotates the object. As such, if you orient a pump by setting a particular nozzle axis parallel to a nozzle axis on a different equipment object, only the orientation of the equipment is affected, not its location. This option is not available for Designed Equipment.

In the following illustration, the motor of the object on the right is selected as an input for the parallel constraint, and the motor of the object on the left is selected as the reference. The result is that the right-hand object is rotated (but not moved) so that the two motors are parallel.



See Also

[Rotate Equipment \(on page 131\)](#)

[Place Equipment from the Catalog \(on page 22\)](#)

[Replace Equipment \(on page 134\)](#)

[Set Positioning Relationships for Equipment \(on page 23\)](#)

Assigning Part Names at Placement

Every task that creates new parts in the model must assign a part name to each new part.

In the Equipment and Furnishings task, the part name is automatically generated using the default name rule when you place the part. The name consists of the part number of the equipment and a random number. The part number is defined on the part class worksheet in **Equipment.xls**.

You can change the part name on the *Occurrence Tab (Nozzle Properties Dialog Box)* (on page 111) for each part. When you override the automatically generated part name in the **Name** box on the **Occurrence** tab, the text in the **Rule** box changes to **User Defined**.

See Also

Equipment Properties Dialog Box (on page 27)

Place Equipment from the Catalog (on page 22)

Replace Equipment (on page 134)

Create an Equipment System

1. Click **Tasks > Systems and Specifications**.
2. In the tree view, select the system in which to create the equipment system.
3. From the ribbon, select **New Equipment System** 
4. Select the new system in the tree view, and then right-click and select **Properties**.
5. Change any properties as needed.

 **NOTE** You can right-click in the **Workspace Explorer** and select **Create System** from the menu, instead of starting Systems and Specifications.

Select Equipment Dialog Box

Specifies the equipment needed for placement. This dialog box appears automatically when you click the **Place Equipment**  command. By browsing through the part hierarchy, you can find any piece of equipment in the Catalog database. After you select a piece of equipment and click **OK**, the **Equipment Properties** dialog box appears so that you can define properties for the new equipment. After you click **OK** on the **Properties** dialog box, the software returns you to the model so that you can finalize configuration and placement.

Save

This option is unavailable.

Cut

This option is unavailable.

Copy

This option is unavailable.

Paste

This option is unavailable.

Delete

This option is unavailable.

Refresh

This option is unavailable.

Insert Row

This option is unavailable.

Move Up

This option is unavailable.

Move Down

This option is unavailable.

Properties

Displays the equipment properties as defined in the catalog.

Preview

Displays a bitmap symbol of the selected equipment. The image file must be assigned to the equipment in the catalog reference data.

Filter

This option is unavailable.

Sort

This option is unavailable.

Customize Current View

This option is unavailable.

List View

Sets the dialog box to display equipment in a list view.

Grid View

Sets the dialog box to display equipment in a spreadsheet-style grid view.

Back

Returns you to the previously selected equipment part or node. Use this command to navigate through the equipment hierarchy to the specific part you need.

Forward

Sends you to the last selected equipment part or node that you moved away from by using the **Back** button. Use this command to navigate through the equipment hierarchy to the specific part you need.

Up One Level

Brings up the next highest level of the Equipment catalog hierarchy. Use this command to navigate through the equipment hierarchy to the specific part you need.

Check Data

This option is unavailable.

Check Data Ignored Inconsistencies

This option is unavailable.

Help

Displays on-line help for Equipment and Furnishings.

Address

Specifies your exact location within the displayed hierarchy.

SECTION 3

Place Equipment

 Specifies any piece of equipment from the Equipment folder of the catalog and places an occurrence of it inside the model. You can modify the offset of the equipment, its relationships to other equipment, and other properties during or after placement. Using positioning relationships, the **Place Equipment** command allows you to mate, connect, or align equipment, and you can use common tools like the **PinPoint** command for precise positioning of the equipment.

If a parent object is selected when you start **Place Equipment**, the **System** property is automatically updated with the parent object.

You place equipment objects into the model by selecting the equipment from the **Select Equipment** dialog box and positioning the equipment in the model using the available relationships. When you select an equipment object from the Catalog, you can define a default surface so that, when the equipment is placed into the model, the software automatically creates a relationship to any other surface or reference element that you select. For example, you might define the bottom of a pump as the default surface because you want the software to mate that surface to the floor. If the software cannot find a suitable mating surface for the equipment, the equipment object is placed in free space, pending additional design of the structures or other reference elements.

You can also provide an optional offset distance from the surface or reference elements. The default offset distance for any new piece of equipment is zero or the last offset used in the current session. The software maintains the offset relationship between the default surface and the surface or reference element in the workspace in the event of any changes to their position (for example, if a slab is lowered, then the equipment follows the slab, maintaining the offset). You can further define the equipment position and orientation by mating, aligning, or connecting equipment to other objects in the model, or by moving or rotating the equipment.

In addition to placing equipment from the **Select Equipment** dialog box, you can also drag and drop equipment directly from the **Catalog Browser** in another session opened on the same computer. In this case, the software places the equipment in free space, with relationship to other elements in the model made by locating objects while dragging the equipment in the new session.

Cable and Equipment

If you are going to use an object for routing cable, you must not model that object as equipment.

Although it may sometimes be desirable to model a piece of raceway, cable tray, or cable bus as a piece of equipment, you should not do this if there is any possibility of ever needing to route cable through the object. This is because cable routes cannot traverse equipment; they can only be terminated by equipment. A cable cannot be properly routed through a piece of equipment. The cable can only interconnect two pieces of equipment.

An example is a pull box. This object should be created in the catalog data as a conduit fitting, not as a piece of equipment. Otherwise, the object will render conduit networks useless for cable routing every time it is used.

Equipment with Occurrence Properties Modeled Using Solid Edge

You cannot place equipment with occurrence properties modeled using Solid Edge unless Solid Edge is installed on your computer. If you try to place such an object, the following message appears: **Cannot start Solid Edge. Exiting InitializSymbolDefinition.**

As a workaround to the Solid Edge requirement, you can use the Bulkload utility with the flavors option. Designers can create dozens of variations for any equipment part imaginable. Also, by creating several variations of a part, rather than using occurrence properties to create the variations automatically, designers can implement custom Solid Edge equipment without having to install Solid Edge on every designer's computer.

Each variation of a part represents some minor deviation from the catalog part, be it on the basis of size, operational specifications, or material. With Solid Edge, new variations can be created on the basis of size, while the software allows manipulation of the operating parameters or material of construction in the reference data. For more information on bulkloading with flavors, refer to the *Reference Data Guide*, accessible from the **Help > Printable Guides** command in the software.

Place Equipment Ribbon

Sets options for adding equipment to your model. This ribbon appears automatically after you select the **Place Equipment**  command and then select an equipment object, or when you select an existing piece of equipment.

TIP To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.

Equipment Properties

Edits the occurrence properties and reviews the static properties of an existing piece of equipment. Equipment properties can be set only after an equipment object is placed in the model. Equipment properties can be edited only after the object is placed in the model.

NOTE Any object modeled in Solid Edge that has occurrence properties cannot be placed or modified unless your computer has a copy of Solid Edge installed.

User Defined Form Definition

Displays **User Defined Form** for the particular equipment if it is defined in the catalog. See the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Relationship List

Lists all relationships for the selected equipment and provides an option for creating a new relationship if the equipment is not already fully constrained. An equipment part is fully constrained when it has sufficient defined relationships to prevent movement or rotation of the part along all three coordinate axes.

Positioning Relationships

Displays the available options for types of positioning relationships. Some options may not be available for all equipment types. See *Positioning Relationships* (on page 12) for more information.

Delete Relationship

Removes the selected relationship from the equipment model and the database. Using the **Relationships** list box, select a previously existing relationship for the equipment, and click

Delete Relationship. You can use this command only when modifying existing equipment.

 **Equipment Reference**

Prompts you for the reference on the equipment to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

 **Second Part Reference**

Prompts you for the reference on the equipment object or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first equipment part chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment placement if needed.

Name

Displays the equipment name, as dictated by your predefined name rules, and accepts changes to that name.

System

Specifies the system with which to associate the selected equipment. The default system is the Model itself.

<Select Graphically>

Allows you to select the system from the graphic view or from the **Workspace Explorer**. You can then place or modify the equipment.

Select Equipment Dialog Box

Specifies the equipment needed for placement. This dialog box appears automatically when you click the **Place Equipment**  command. By browsing through the part hierarchy, you can find any piece of equipment in the Catalog database. After you select a piece of equipment and click **OK**, the **Equipment Properties** dialog box appears so that you can define properties for the new equipment. After you click **OK** on the **Properties** dialog box, the software returns you to the model so that you can finalize configuration and placement.

 **Save**

This option is unavailable.

 **Cut**

This option is unavailable.

 **Copy**

This option is unavailable.

 **Paste**

This option is unavailable.

 **Delete**

This option is unavailable.

 **Refresh**

This option is unavailable.

 **Insert Row**

This option is unavailable.

 **Move Up**

This option is unavailable.

 **Move Down**

This option is unavailable.

 **Properties**

Displays the equipment properties as defined in the catalog.

 **Preview**

Displays a bitmap symbol of the selected equipment. The image file must be assigned to the equipment in the catalog reference data.

 **Filter**

This option is unavailable.

 **Sort**

This option is unavailable.

 **Customize Current View**

This option is unavailable.

 **List View**

Sets the dialog box to display equipment in a list view.

 **Grid View**

Sets the dialog box to display equipment in a spreadsheet-style grid view.

 **Back**

Returns you to the previously selected equipment part or node. Use this command to navigate through the equipment hierarchy to the specific part you need.

 **Forward**

Sends you to the last selected equipment part or node that you moved away from by using the **Back** button. Use this command to navigate through the equipment hierarchy to the specific part you need.

 **Up One Level**

Brings up the next highest level of the Equipment catalog hierarchy. Use this command to navigate through the equipment hierarchy to the specific part you need.

Check Data

This option is unavailable.

Check Data Ignored Inconsistencies

This option is unavailable.

Help

Displays on-line help for Equipment and Furnishings.

Address

Specifies your exact location within the displayed hierarchy.

What do you want to do?

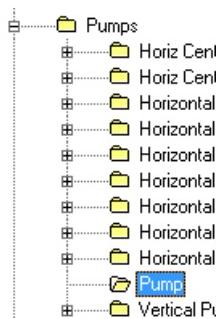
- *Place equipment from the catalog* (on page 22)
 - *Set positioning relationships for equipment* (on page 23)
 - *Place doors from the catalog* (on page 24)
 - *Place windows from the catalog* (on page 25)
 - *Edit equipment properties* (on page 26)
 - *Edit equipment relationships* (on page 27)
-

Place equipment from the catalog

1. Click **Place Equipment**  on the vertical toolbar.

The **Select Equipment** dialog box displays.

2. Locate the necessary equipment type using the tree view. Expand the nodes for the general type of equipment that you need, continuing until a list of available parts appears in the catalog window.



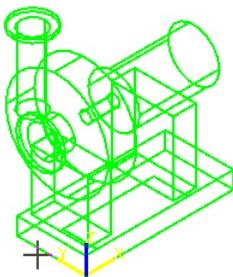
- In the list view, select the equipment object.

Name	Part Description	Symbol Definition	Definition
PUMP_001A-E	Centrifugal Pum...	SP3DPumpAsm....	SP3DPumpAsm..
PUMP_001A_IMP-E	Centrifugal Pum...	SP3DPumpAsm....	SP3DPumpAsm..
PUMP_001A_OIL-E	Centrifugal Pump	SP3DPumpAsm....	SP3DPumpAsm..

- Click **OK** on the **Select Equipment** dialog box.

The **Equipment Properties** dialog box displays so that you can define properties for the new object.

- Define properties as needed, and then click **OK** to return to the workspace and place the equipment. For more information, see *Equipment Properties Dialog Box* (on page 27).
- Click in a graphic view to select the mounting surface and approximate position for the object.



TIP Press the left and right arrow keys to rotate the equipment by 90-degree increments at any time during the placement of the equipment. Press the up arrow key to scroll through the three possible axes of rotation. Press the down arrow key to scroll to each datum point.

- Using the **Place Equipment** ribbon, do any of the following, if needed:
 - Set an offset for the relationship by specifying the distance in the **Offset** box.
 - Click **Properties** on the ribbon, and type any necessary property information in the grid provided on the **Equipment Properties** dialog box.
 - Add or change a positioning relationship by selecting it from the **Relationship** list on the **Place Equipment** ribbon.
 - Select a system with which to associate the new object in the **System** box.

NOTES If you intend to use an object for routing cable, you must not model that object as equipment. For example, a pull box should be created in the catalog data as a conduit fitting, not a piece of equipment. Otherwise, the object will render conduit networks useless for cable routing every time it is used.

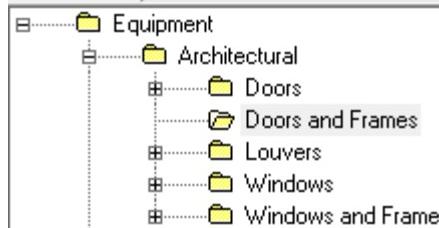
Set positioning relationships for equipment

- Select the equipment for which you need to define a relationship.
- In the **Relationship List** box on the **Place Equipment** ribbon, select <New Relationship>.
- Select one of the available positioning relationships in the **Positioning Relationships** list box, and follow the status bar prompts.

NOTE Depending on the type of positioning relationship defined in the previous step, surfaces or points need to be defined to finish configuring the relationship. For more information, see *Positioning Relationships* (on page 12).

Place doors from the catalog

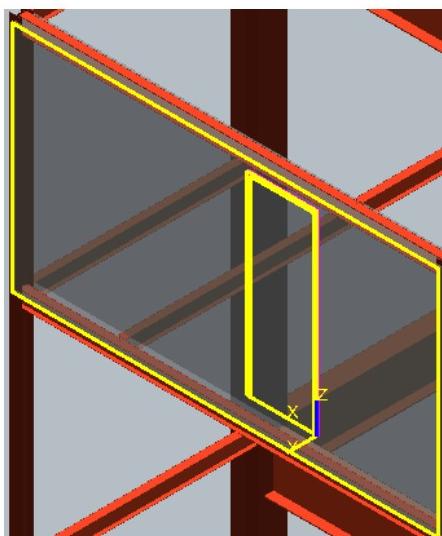
1. Click **Place Equipment**  on the vertical toolbar.
- The **Select Equipment** dialog box displays.*
2. Locate the door to place under the Equipment > Architectural node.



3. In the list view, select the door to place.

Name	Part Description	Symbol Definition	Definition
Simple_Centered...	Swing Door Cen...	SimpleDoorAsm....	SimpleDoorAsm....
Simple_Flushed ...	Roll Up Door Flu...	SimpleDoorAsm....	SimpleDoorAsm....
Simple_At distanc...	Roll Up Door at ...	SimpleDoorAsm....	SimpleDoorAsm....

4. Click **OK** on the **Select Equipment** dialog box.
- The **Equipment Properties** dialog box displays so that you can define properties for the new object.*
5. Define properties as needed, and then click **OK** to return to the workspace and place the equipment. For more information, see *Equipment Properties Dialog Box* (on page 27).
6. Click in a graphic view to select the mounting surface and approximate position for the object.

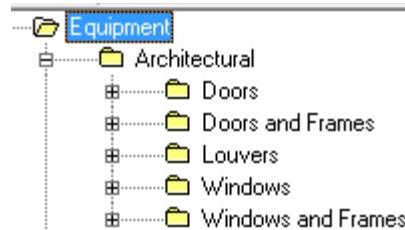


TIP Press the left and right arrow keys to rotate the door by 90-degree increments at any time during the placement of the door. Press the up arrow key to scroll through the three possible axes of rotation.

7. Using the **Place Equipment** ribbon, do any of the following, if needed:
 - a. Set an offset for the relationship by specifying the distance in the **Offset** box.
 - b. Click **Properties**  on the ribbon, and type any necessary property information in the grid provided on the **Equipment Properties** dialog box.
 - c. Add or change a positioning relationship by selecting it from the **Relationship** list on the **Place Equipment** ribbon.
 - d. Select a system with which to associate the new object in the **System** box. We recommend that you place the door in the same system as the parent wall.

Place windows from the catalog

1. Click **Place Equipment**  on the vertical toolbar.
*The **Select Equipment** dialog box displays.*
2. Locate the window to place under the Equipment > Architectural > Windows node.

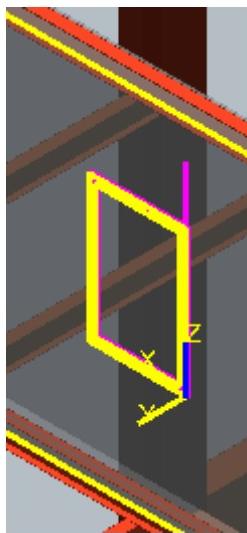


3. In the list view, select the window part to place.

Name	Part Description	Symbol Definition	Definition
Simple_Centered...	Swing Window ...	SimpleDoorAsm....	SimpleDoorAsm....
Simple_Flushed_...	Swing Window fl...	SimpleDoorAsm....	SimpleDoorAsm....
Simple_AtDistanc...	Swing Window ...	SimpleDoorAsm....	SimpleDoorAsm....

4. Click **OK** on the **Select Equipment** dialog box.
*The **Equipment Properties** dialog box displays so that you can define properties for the new object.*
5. Define properties as needed, and then click **OK** to return to the workspace and place the equipment. For more information, see *Equipment Properties Dialog Box* (on page 27).

6. Click in a graphic view to select the mounting surface and approximate position for the object.



TIP Press the left and right arrow keys to rotate the window by 90-degree increments at any time during the placement of the window. Press the up arrow key to scroll through the three possible axes of rotation.

7. Using the **Place Equipment** ribbon, do any of the following, if needed:
 - a. Set an offset for the relationship by specifying the distance in the **Offset** box.
 - b. Click **Properties**  on the ribbon, and type any necessary property information in the grid provided on the **Equipment Properties** dialog box.
 - c. Add or change a positioning relationship by selecting it from the **Relationship** list on the **Place Equipment** ribbon.
 - d. Select a system with which to associate the new object in the **System** box. We recommend that you place the window in the same system as the parent wall.

Edit equipment properties

1. Select the equipment object that you need to modify.
2. Click **Equipment Properties**  on the horizontal ribbon.

The **Equipment Properties** dialog box displays.

TIP You can also access the **Equipment Properties** dialog box by selecting **Properties** from the **Edit** menu, or by right-clicking the selected object and choosing **Properties** from the short-cut menu.

3. Choose the appropriate dialog box tabs, and modify the properties as needed.
4. Click **OK** to save your changes, and return to the workspace.

TIP You can also select **Apply** to put the changes into effect and continue working in the **Equipment Properties** dialog box.

NOTES

All occurrence and definition properties for an equipment object are defined by the Microsoft Excel® workbook named **Equipment.xls** in the reference data. For information on adding or editing different types of properties, refer to the *Reference Data Guide*, accessible from the **Help > Printable Guides** command.

- If more than one piece of equipment is selected, the **Occurrence** tab displays only the common occurrence properties of the objects.
- The **Definition**, **Connections**, **Weight & CG**, and **Relationship** tabs are read-only. You cannot edit this information from the **Equipment Properties** dialog box.
- If the equipment model was created using Solid Edge, then any changes that affect the shape or size of the equipment model require that you have Solid Edge running on your computer.

Edit equipment relationships

1. Select the equipment to modify.

 **TIP** To edit the properties of a designed equipment object for which geometry has yet to be defined, you must select the object from the system hierarchy in the **Workspace Explorer**.

2. Specify the relationship in the **Relationship List** box.
3. Change the type of relationship using the **Positioning Relationship** box as needed.

Set positioning relationships for equipment (on page 23)

Set positioning relationships for designed equipment (on page 42)

 **NOTE** You can remove relationships by clicking **Delete Relationship**  on the ribbon. This step is often necessary when moving previously constrained equipment. After a relationship is deleted, no connectivity remains between the two pieces of equipment during further design operations.

Equipment Properties Dialog Box

Displays equipment properties for review and editing.

See Also

Occurrence Tab (Equipment Properties Dialog Box) (on page 27)

Definition Tab (Equipment Properties Dialog Box) (on page 36)

Connection Tab (on page 37)

Configuration Tab (on page 148)

Notes Tab (on page 150)

Edit Equipment Properties (on page 26)

Edit Equipment Relationships (on page 27)

Occurrence Tab (Equipment Properties Dialog Box)

Displays all the editable instance-specific information about the selected equipment object. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If more than one equipment object is selected, only the common occurrence properties for the selected objects appear on the tab.

When viewing properties for a single equipment object, the following properties appear. More properties may appear depending on what you defined in the reference data. For more information about occurrences defined in the reference data, see the *Equipment and Furnishings Reference Data Guide* available with the **Help > Printable Guides** command in the software.

NOTE Because equipment properties are customizable in the equipment and furnishings reference data, only the properties that are required by the software are documented.

Show Dimensional Legend

Displays the bitmap image associated with the equipment object if it has been defined in the reference data. The image displays in a separate window.

Category

Select the properties to view, modify, or define. Equipment properties are divided into several different categories: **Standard**, **Position and Orientation**, **Insulation and Tracing**, **Weight and CG**, **Equipment Dimension**, **Fabrication and Construction**, **Surface Treatment and Coating**, and **Responsibility**.

Standard

Name

Displays the name of the equipment object. The equipment name is based on the **Name Rule** selection. If you type a name in this field, the **Name Rule** property updates to **User Defined**.

Name Rule

Specify the naming rule to use to name this equipment object. You can select one of the listed rules or select **User Defined** to specify the equipment name yourself in the **Name** box.

Description

Specifies a description for the object.

Equipment

Select the system to which the object belongs. By default, the model is the parent system for the object. If a parent system is selected when you start the command, **System** is automatically updated with the selected system. If multiple systems are selected when you start the command, **System** is automatically updated with the last used system.

Reporting Requirements

Specifies the reporting requirement for the object. To change the options on the list, edit the **Reporting Type** select list in Catalog.

Reporting Type

Specifies the type of reporting. To change the options on the list, edit the **Reporting Type**

select list in Catalog.

Behavior Controlled by User

Indicates whether or not you can delete the object. Select **False** to prevent anyone from deleting the object from the model.

Correlation Status

Specifies whether or not the object has been correlated to an object in a P&ID. The list is defined by the EFWCorrelationStatus select list.

Correlation Basis

Specifies whether or not the object is correlated to a P&ID object. The list is defined by the EFWCorrelationBasis select list.

Correlate Object indicates that the object has a correlating object in a P&ID.

No correlation is required indicates that the object does not have a correlating object in a P&ID.

Correlation Approval Status

Specifies whether or not the object is approved with discrepancies in the three-dimensional data compared with design basis data. The list is defined by the EFWCorrelationApprovalStatus sheet in the AllCodelist workbook.

Topology mismatch approved indicates that the object topology mismatch can be ignored.

Data and Topology mismatches approved indicates that the object data and topology mismatches can be ignored.

None indicates that you do not approve a mismatch.

Position and Orientation

East

Displays the distance of the connection point from the active coordinate system origin in the east direction.

North

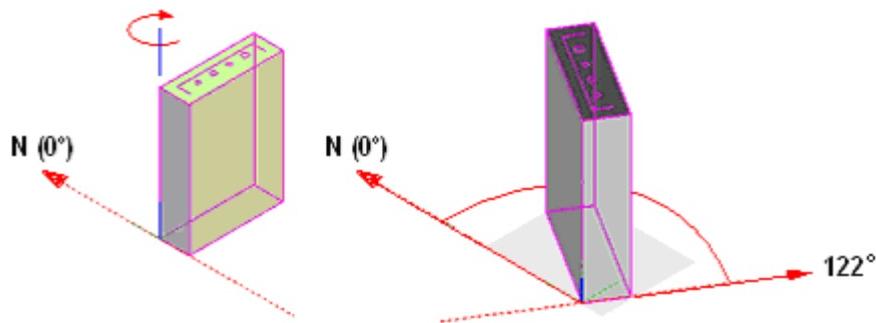
Displays the distance of the connection point from the active coordinate system origin in the north direction.

Elevation

Displays the distance of the connection point above or below the active coordinate system origin.

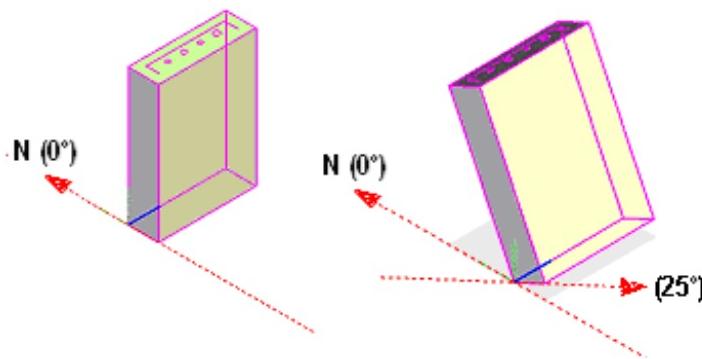
Bearing

Specifies the bearing angle for the object. The **Bearing** angle is measured between the local x-axis of the object and the Y-axis (North) of the global coordinate system in the XY-plane. The local x-axis is the default axis of primary symmetry for all symbols in the catalog. The bearing measurement direction is clockwise from the active coordinate system North looking in the negative active coordinate system direction; that is, down from 0 to 360 degrees. You can enter negative bearing angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the bearing measure displays 0 degrees.



Pitch

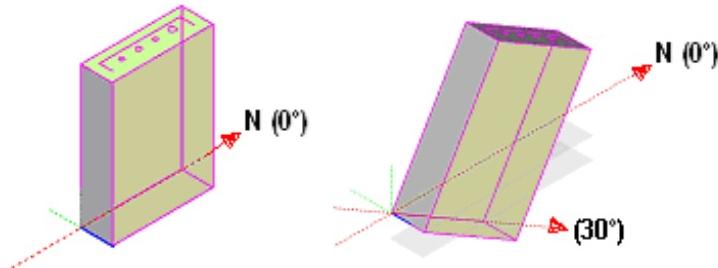
Specifies the pitch angle for the shape. The **Pitch** angle measures between the x-axis of the object and the X-axis of the global coordinate system in the XZ-plane. This option sets the reference in the model to a line that is the intersection of the vertical plane through the X-axis of the local coordinate system and the active coordinate system horizontal plane. The angle is measured in the positive direction from the horizontal plane in the active coordinate system up direction regardless of the current bearing. Another way to describe pitch is the rotation of the object about its y-axis. Pitch angles are limited to between -90 degrees and +90 degrees, with 0 indicating horizontal.



Roll

Specifies the roll angle for the object. The **Roll** angle measures between the local z-axis of the object and the Z-axis of the global coordinate in the YZ-plane. Another way to describe roll is the rotation of the object about its x-axis. This option sets the reference in the model to a line that is perpendicular to the local coordinate system x-axis and in the horizontal plane. Roll angles are measured clockwise from horizontal to the y-axis of the local coordinate system. The roll angle is between 0 and 360 degrees. You can enter negative roll angles, but the software automatically converts them to the positive equivalents. If the pitch is set to

+/- 90 degrees, then the reference in the model is the North axis.



Insulation and Tracing

Is Insulated

Indicates whether or not the object is insulated.

Thickness

Specifies the thickness of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Thickness** box. However, if **Insulation Specification** is manually defined, select the insulation thickness from those values allowed for the material in the reference data. If the value in the **Insulation Specification** field is a selected reference data insulation specification, the software reads the thickness from the insulation specification from the reference data and displays it here.

Requirement

Specifies the requirement of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Insulation Type

Specifies the type of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Insulation Purpose

Displays the purpose of the insulation. If **Insulation Specification** is set to use the insulation defined by the equipment object, the information in this box cannot be changed. However, if **Insulation Specification** is manually defined, select the purpose of the insulation in this box. Available purposes are read from the reference data.

Material

Specifies the material of the insulation. If **Insulation Specification** is set to use the insulation defined by the equipment object, this field cannot be changed. However, if **Insulation Specification** is manually defined, select the insulation material from those available from the reference data.

The list contains the last five insulation materials selected. Click **More** to browse the catalog for the insulation material to use. If you set this field to **Not Insulated**, the **Insulation Thickness** field is disabled. If you set this field to **Use Run Material**, the along leg feature inherits the insulation material from the duct run.

To change the options on the list, edit the **Insulation Material** select list in Catalog.

Operating Temperature

Specifies the operating temperature.

Insulation Surface Area

Specifies the measurement of the surface area of insulation.

Heat Tracing Requirement

Indicates whether or not the equipment is heat-traced. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Type

Specifies the type of heat-tracing. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium

Specifies the heat-tracing medium to apply to the object. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium Temperature

Specifies the temperature of the heat-tracing medium. Include the unit of measurement of temperature (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius, for example).

Weight & CG

Displays the center-of-gravity and the weight of the selected equipment objects. The center-of-gravity locations are displayed in global system coordinates along the X-, Y-, and Z-axes. The software includes the insulation weight in the calculated weight. If you specify the weight yourself, you must include the insulation weight value.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

 **NOTE** For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CGX

Specifies the X-axis location of the dry center-of-gravity.

Dry CGY

Specifies the Y-axis location of the dry center-of-gravity.

Dry CGZ

Specifies the Z-axis location of the dry center-of-gravity.

Wet CGX

Specifies the X-axis location of the wet center-of-gravity.

Wet CGY

Specifies the Y-axis location of the wet center-of-gravity.

Wet CGZ

Specifies the Z-axis location of the wet center-of-gravity.

Equipment Dimension**Property**

Displays the name of the property. The properties that appear are dependent on the equipment type selected. For example, the properties displayed for an equipment object based on a pump are different from those based on a cooler. For more information on equipment part properties, see the *Equipment and Furnishings Reference Data Guide*, available with the **Help > Printable Guides** command within the **Equipment and Furnishings** task.

Value

Displays the value of the corresponding property.

Fabrication and Construction**Fabrication Requirement**

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating**Exterior Coating Requirement**

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility**Cleaning Responsibility**

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

See Also

Equipment Properties Dialog Box (on page 27)

Select Insulation Material Dialog Box

Specifies the insulation materials for a selected object. By browsing through the part hierarchy, you can find and select an insulation material in the Catalog database.



This option is unavailable.

Cut

This option is unavailable.

 **Copy**

This option is unavailable.

 **Paste**

This option is unavailable.

 **Delete**

This option is unavailable.

 **Undo**

This option is unavailable.

 **Insert Row**

This option is unavailable.

 **Move Up**

This option is unavailable.

 **Move Down**

This option is unavailable.

 **Properties**

Displays the properties of the selected object. Because you cannot modify any properties until the object is placed, all properties on the **Properties** dialog box are read-only.

 **Preview**

Displays a bitmap symbol of the selected object. The image file must be assigned to the object in the catalog reference data.

 **Filter**

This option is unavailable.

 **Sort**

This option is unavailable.

 **Customize Current View**

This option is unavailable.

 **List View**

Sets the dialog box to display insulation thickness values in a list view.

 **Grid View**

Sets the dialog box to display insulation thickness values in a spreadsheet-style grid view.

 **Back**

Returns you to the previously selected insulation material folder. Use this command to navigate through the hierarchy to the specific insulation material you need.

 **Forward**

Sends you to the last selected insulation material folder that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific insulation material you need.

 **Up One Level**

Brings up the next highest level of the hierarchy. Use this command to navigate through the hierarchy to the specific material you need.

 **Check Data**

This option is unavailable.

 **CheckData Ignored Inconsistencies**

This option is unavailable.

 **Smart 3D Help**

This option is unavailable.

Address

Sets the current location within the hierarchy of available insulation materials. The dropdown box lists the folders you have visited.

Definition Tab (Equipment Properties Dialog Box)

Displays the part information for the equipment object, the properties and their values, as defined in the reference data. If more than one equipment object is selected, only the common properties for the selected objects display on the tab. For more information about the information defined in the reference data, see the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Category

Select the properties to review. Equipment information is divided into several different categories, such as **Standard**, **Equipment Specification**, **Equipment Support**, **Nozzle Length**, and **Equipment Dimensions**.

 **IMPORTANT** Not all categories are available for all equipment objects.

Property

Displays the name of the property. The properties that appear are dependent on the equipment type selected. For example, the properties displayed for an equipment object based on a pump are different from those based on a cooler. For more information on equipment part properties, see the *Equipment and Furnishings Reference Data Guide*, available with the **Help > Printable Guides** command within the Equipment and Furnishings task.

Value

Displays the value of the corresponding property.

Connection Tab

Displays information on the connection points of a piece of equipment, including piping, electrical, foundation, or HVAC connections, and Reference 3D objects. If more than one equipment object is selected, only the common properties of the connections for the selected objects display on the tab. For more information about the information defined in the reference data, see the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Connector

Select the connector for which to view properties. With the exception of Reference 3D object connectors, the connectors display in alphabetical order. When you select a connector from the list, the graphic object associated with the connection, if one exists, highlights in the graphic view for further visual confirmation of the port selected.

Property

Lists all the properties of the selected connection. These properties vary depending on the type of connection selected.

Value

Displays the value of the corresponding property.

Insulation Tab

Displays insulation properties that have been defined for the selected equipment.

Insulation Specification

Specifies whether or not the selected equipment object is insulated. The options on this dialog box remain disabled unless **User Defined** is selected from the dropdown list.

Purpose

Specifies the insulation purpose for the selected equipment object.

Material

Displays the **Select Insulation Material** dialog box from which you can specify an insulation material. For more information, see *Select Insulation Material Dialog Box* (on page 34).

Thickness

Defines the thickness of the specified insulation material. This option remains disabled until a material is selected from the **Select Insulation Material** dialog box. By default, the smallest thickness value of the selected material is displayed. An alternative value may be selected from the dropdown list.

See Also

Designed Equipment Properties Dialog Box (on page 44)
Equipment Properties Dialog Box (on page 27)

SECTION 4

Place Designed Equipment

 Places equipment types that have been defined in the reference data. The properties of the equipment type you select are inherited by the designed equipment part. With the **Place Designed Equipment** command, you can define an equipment type, a system parent, name, and other property values as appropriate for the equipment type. Equipment position and orientation is further defined by mating, aligning, or connecting equipment graphics to reference graphics in the workspace, or by moving or rotating the equipment.

The software's designed equipment component modeling capabilities allow you to build an equipment definition in the Model database by combining basic shapes, ports (nozzle, foundation, electrical, and so forth) and properties defined in the reference data. Designed equipment objects can be built entirely out of primitive geometric shapes, designed equipment components, or an existing equipment component part can be placed from the catalog to enhance the designed equipment object. For example, you can use the **Place Designed Equipment**  command to create a designed equipment object in the model, add an agitator (using the **Place Shape**  command), and then add nozzle primitives (using the **Place Nozzle**  command) to customize the designed equipment. You can also place a nozzle using a nozzle defined in the P&ID design basis (if P&ID design basis data is available).

Some equipment shapes can be modeled more easily using solid modeling software. In addition to using the primitive shapes delivered with the software, you can import graphics from an SAT file as the geometry for the designed equipment shape. After the geometry is defined, you can then place ports to define distributed connections to the designed equipment.

As a further enhancement, the software provides a prismatic shape feature so that you can design uniquely customized shapes in the model. The place prismatic shape feature is made up of two distinct processes:

- Define a path.
- Define a cross section, or profile, to project along the path.

The ultimate goal of these two processes is placement of a shape in the model as part of a designed equipment object.

After a path and cross-section have been defined, the cross- section is then projected along the path to create a shape with the same properties as other equipment shapes.

 **NOTE** The place prismatic shape feature is available on the **Occurrence** tab of the **Shape Properties** dialog box.

When you create a designed equipment object and add an equipment component object or shape, the default surface on the equipment component object or shape as defined in Visual Basic is automatically used to establish a mate or align relationship to the other surface or reference element you select during initial placement. When modifying a shape to create a new mate or align relationship, the default surface is not automatically used. You must interactively select the surface. For example, you might define the bottom of a pump as the default surface so that the software mates that surface to the floor. If the software cannot find a suitable mating surface for the equipment, the equipment object is placed in free space, pending additional design of the structures or other reference elements.

When creating a designed equipment object, you can provide an optional offset distance from the surface. The default offset distance for any new piece of equipment is zero or the last offset used in the current session. The software maintains the offset relationship between the default surface and the surface or reference element in the workspace in the event of any changes to their position. For example, if a slab is lowered, then the equipment follows the slab, maintaining the offset. You can further define the equipment orientation by mating, aligning, or connecting equipment to other objects in the model, or by moving or rotating the designed equipment. For more information on positioning relationships, see *Positioning Relationships* (on page 12).

Place Designed Equipment Ribbon

Sets options for adding designed equipment to your model. This ribbon appears automatically after you select the **Place Designed Equipment**  command and then select an equipment type. Until a shape has been added to the designed equipment, only the **Equipment Properties**, **Equipment Name**, and **Active System** controls are available.

TIPS

- To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.
- To add a shape to the designed equipment and enable the remaining ribbon controls, select **Place Shape** .

Equipment Properties

Edits the occurrence properties and review the static properties of an existing piece of equipment. Equipment properties can be edited only after the designed equipment is placed in the model.

User Defined Form Definition

Displays **User Defined Form** for the particular equipment if it is defined in the catalog. See the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Relationship List

Lists all relationships for the selected designed equipment and provides an option for creating a new relationship if the equipment is not already fully constrained. An equipment object is fully constrained when it has sufficient defined relationships to prevent movement or rotation of the object along all three coordinate axes. This control is only available after a shape has been added to the designed equipment.

Positioning Relationships

Displays the available options for types of positioning relationships: Mate, Align, Connect Minimum Distance, Mate to Tangent Plane, and Parallel. Some options may not be available for all designed equipment types. See *Positioning Relationships* (on page 12) for more information.

Delete Relationship

Removes the selected relationship from the equipment model and the database. Using the **Relationships List** box, select a previously existing relationship for the designed equipment, and click **Delete Relationship**. You can use this command only when modifying existing designed equipment. This control is available only after a shape has been added to the designed equipment.

Equipment Reference

Prompts you for the reference on the designed equipment to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

Second Part Reference

Prompts you for the reference on the equipment object or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first designed equipment object chosen with respect to the second object selected in the definition of the relationship. This control is available only after a shape has been added to the designed equipment.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment placement if needed. This control is available only after a shape has been added to the designed equipment.

Name

Displays the designed equipment name, as dictated by your predefined name rules, and accepts changes to that name.

System

Specifies the system with which to associate the selected equipment. The default system is the Model itself.

Select Equipment Type Dialog Box

Selects the type of designed equipment to be created. This dialog box appears automatically when you click the **Place Designed Equipment**  command. By browsing through the part hierarchy, you can find any piece of equipment in the Catalog database. The resulting designed equipment inherits the properties of the existing equipment type you select from the dialog box. After you select an equipment type and click **OK**, the **Properties** dialog box appears so you can set properties for the new equipment. After you click **OK** on the **Properties** dialog box, the software returns you to the model so you can finish placing the object.

Back

Returns you to the previously selected equipment type or node. Use this command to navigate through the equipment hierarchy to the specific type you need.

Forward

Sends you to the last selected equipment type or node that you moved away from by using the **Back** button. Use this command to navigate through the equipment hierarchy to the specific type you need.

Up One Level

Brings up the next highest level of the Equipment catalog hierarchy. Use this command to navigate through the equipment hierarchy to the specific type you need.

Properties

Displays the properties of the selected object. Because you cannot modify any properties until the equipment is placed, all properties on the **Properties** dialog box are read-only.

Preview

Displays a bitmap symbol of the selected equipment. The image file must be assigned to the equipment in the catalog reference data.

List View

Sets the dialog box to display equipment in a list view.

Grid View

Sets the dialog box to display equipment in a spreadsheet-style grid view.

Address

Specifies your exact location within the displayed hierarchy.

What do you want to do?

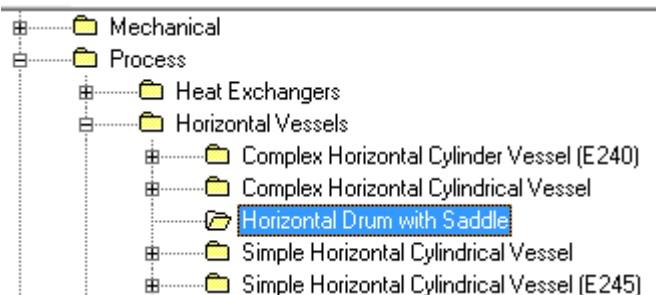
- *Place designed equipment* (on page 41)
- *Set positioning relationships for designed equipment* (on page 42)
- *Edit designed equipment properties* (on page 42)
- *Edit designed equipment reference coordinate system* (on page 43)

Place designed equipment

1. Click **Place Designed Equipment**  on the vertical toolbar.

The **Select Equipment Type** dialog box displays.

2. Select the designed equipment type to create.



★IMPORTANT The **Select Equipment Type** dialog box displays the existing equipment classifications as defined in the Catalog database. The resulting designed equipment object inherits the properties of the equipment type you select.

3. Click **OK**.

*A definition of the new designed equipment object is created in the Model database and displays in the system hierarchy in the **Workplace Explorer**.*

*The **Designed Equipment Properties** dialog box displays.*

4. Edit the designed equipment properties as needed, and click **OK**.

The software returns you to the model to complete the placement of the designed equipment.

5. Click in the graphic view to define the origin of the equipment.

!TIP You can press the down arrow key to scroll to each data point in the equipment.

6. If necessary, use the **Place Designed Equipment** ribbon to do the following:

- Select a system with which to associate the new designed equipment object from the options in the **System** box.
- Type a name for the new designed equipment object in the **Name** box.

!NOTE Although the new designed equipment object is added to the **Workspace**

Explorer, you must use **Place Shape**  to create geometry for the object. For more information, see *Place Shape* (on page 71).

Set positioning relationships for designed equipment

1. Select the designed equipment object for which to define a relationship.
2. In the **Relationships List** box on the **Place Designed Equipment** ribbon, choose <New Relationship>.
3. Select one of the available positioning relationships in the **Positioning Relationships List** box, and follow the on-screen prompts. For more information, see *Positioning Relationships* (on page 12).

!NOTE Depending on the type of positioning relationship defined in the previous step, surfaces or points need to be defined to finish configuring the relationship.

Edit designed equipment properties

1. On the horizontal ribbon, set the **Locate Filter** box to **Equipment**, and then select an equipment object.

!TIPS

- If the **Locate Filter** box is not displayed on the horizontal ribbon, click **Select**  on the vertical toolbar.
- To edit the properties of a designed equipment object for which geometry has yet to be defined, you must select the object from the system hierarchy in the **Workspace Explorer**.

2. Click **Properties**  on the horizontal ribbon.

*The **Properties** dialog box displays.*

TIP You can also access the **Properties** dialog box by selecting **Properties** from the **Edit** menu, or by right-clicking an equipment object and choosing **Properties** from the short-cut menu.

3. Click the appropriate dialog box tabs, and modify the properties as needed.
4. Click **OK** to save your changes, and return to the workspace.

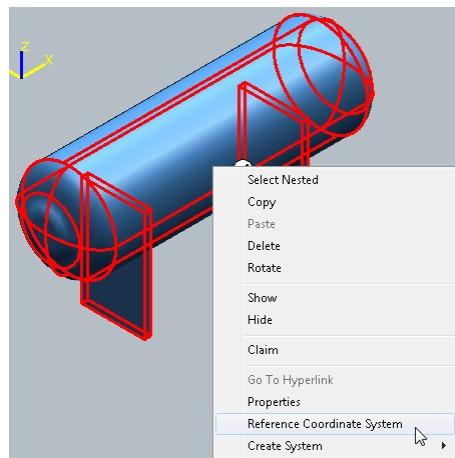
TIP You can also select **Apply** to put the changes into effect and continue working in the **Properties** dialog box.

NOTES

- Occurrence and definition properties for an equipment object are defined by the Excel workbook named **Equipment.xls** in the reference data. For information on adding or editing different types of properties, refer to the *Reference Data Guide*, accessible from the **Help > Printable Guides** command in the software.
- If more than one piece of equipment is selected, the **Occurrence** tab displays only the common occurrence properties of the items.
- The **Definition**, **Connections**, **Weight & CG**, and **Relationship** tabs are read-only. The properties and values displayed on these tabs are defined in the reference data. For more information, see the *Equipment and Furnishings Reference Data Guide*, accessible from the **Help > Printable Guides** command in the software.
- If the equipment model was created using Solid Edge, then any changes that affect the shape or size of the equipment model require that you have Solid Edge running on your computer.

Edit designed equipment reference coordinate system

1. On the horizontal ribbon, set the **Locate Filter** box to **Equipment**, and then select a designed equipment object.
- TIP** If the **Locate Filter** box is not displayed on the horizontal ribbon, click **Select**  on the vertical toolbar.
2. Right-click the selected equipment object, and click **Reference coordinate system**.



3. Click **Move**  to move the reference coordinate system.

The control point of the designed equipment moves with the reference coordinate system.

Designed Equipment Properties Dialog Box

Displays designed equipment properties for review and editing.

See Also

Occurrence Tab (Designed Equipment Properties Dialog Box) (on page 44)

Definition Tab (Equipment Properties Dialog Box) (on page 36)

Configuration Tab (on page 148)

Notes Tab (on page 150)

Occurrence Tab (Designed Equipment Properties Dialog Box)

Displays all the editable instance-specific information about the selected designed equipment object. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If more than one equipment object is selected, common occurrence properties for the selected objects appear on the tab.

When viewing properties for a single equipment object, the following properties appear. More properties may appear depending on what you defined in the reference data. For more information about occurrences defined in the reference data, see the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

 **NOTE** Because equipment properties are customizable in the equipment and furnishings reference data, only the properties that are required by the software are documented.

Display Equipment Preview

This button is disabled for designed equipment.

Category

Select the properties to view, modify, or define. Equipment properties are divided into several different categories: Standard, Insulation and Tracing, Weight and CG, Fabrication and Construction, Surface Treatment and Coating, Position and Orientation, and Responsibility.

Standard

Name

Specifies the name of the object. If a **Name Rule** is specified, then the software uses that rule to determine this name. If the **Name Rule** value is **User Defined**, then you must type a name in this box.

Name Rule

Displays the available name rules for the selected object. Specify the naming rule to use to name the object. You can select one of the listed rules, or you can select **User Defined** to specify the name yourself in the **(Name)** box. **Description**

Specifies a description for the object.

System

Select the system to which the object belongs. By default, the model is the parent system for the object. If a parent system is selected when you start the command, **System** is automatically updated with the selected system. If multiple systems are selected when you start the command, **System** is automatically updated with the last used system.

Reporting Requirements

Specifies the reporting requirement for the object. To change the options on the list, edit the **Reporting Type** select list in Catalog.

Reporting Type

Specifies the type of reporting. To change the options on the list, edit the **Reporting Type** select list in Catalog.

Room Number

Specifies the room number associated with the designed equipment.

Correlation Status

Specifies whether or not the object has been correlated to an object in a P&ID. The list is defined by the EFWCorrelationStatus select list.

Correlation Basis

Specifies whether or not the object is correlated to a P&ID object. The list is defined by the EFWCorrelationBasis select list.

Correlate Object indicates that the object has a correlating object in a P&ID.

No correlation is required indicates that the object does not have a correlating object in a P&ID.

Correlation Approval Status

Specifies whether or not the object is approved with discrepancies in the three-dimensional data compared with design basis data. The list is defined by the EFWCorrelationApprovalStatus sheet in the AllCodelist workbook.

Topology mismatch approved indicates that the object topology mismatch can be ignored.

Data and Topology mismatches approved indicates that the object data and topology mismatches can be ignored.

None indicates that you do not approve a mismatch.

Position and Orientation**East**

Displays the distance of the connection point from the active coordinate system origin in the east direction.

North

Displays the distance of the connection point from the active coordinate system origin in the north direction.

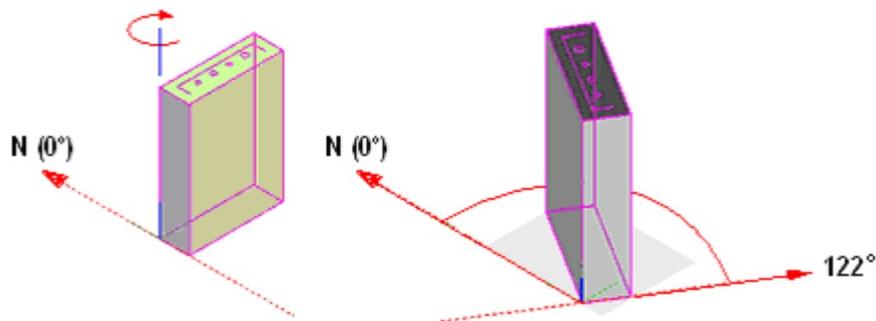
Elevation

Displays the distance of the connection point above or below the active coordinate system

origin.

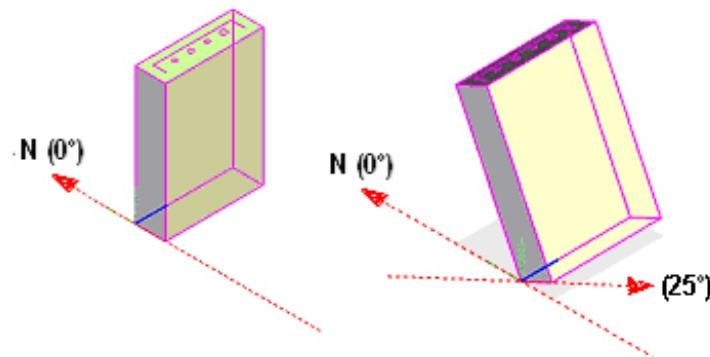
Bearing

Specifies the bearing angle for the object. The **Bearing** angle is measured between the local x-axis of the object and the Y-axis (North) of the global coordinate system in the XY-plane. The local x-axis is the default axis of primary symmetry for all symbols in the catalog. The bearing measurement direction is clockwise from the active coordinate system North looking in the negative active coordinate system direction; that is, down from 0 to 360 degrees. You can enter negative bearing angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the bearing measure displays 0 degrees.



Pitch

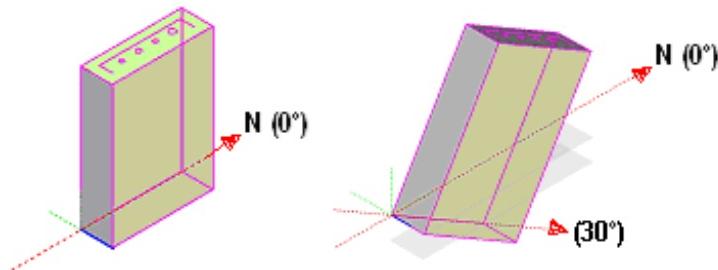
Specifies the pitch angle for the shape. The **Pitch** angle measures between the x-axis of the object and the X-axis of the global coordinate system in the XZ-plane. This option sets the reference in the model to a line that is the intersection of the vertical plane through the X-axis of the local coordinate system and the active coordinate system horizontal plane. The angle is measured in the positive direction from the horizontal plane in the active coordinate system up direction regardless of the current bearing. Another way to describe pitch is the rotation of the object about its y-axis. Pitch angles are limited to between -90 degrees and +90 degrees, with 0 indicating horizontal.



Roll

Specifies the roll angle for the object. The **Roll** angle measures between the local z-axis of the object and the Z-axis of the global coordinate in the YZ-plane. Another way to describe roll is the rotation of the object about its x-axis. This option sets the reference in the model to a line that is perpendicular to the local coordinate system x-axis and in the horizontal plane. Roll angles are measured clockwise from horizontal to the y-axis of the local coordinate.

system. The roll angle is between 0 and 360 degrees. You can enter negative roll angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the reference in the model is the North axis.



Insulation and Tracing

Is Insulated

Indicates whether or not the object is insulated.

Thickness

Specifies the thickness of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Thickness** box. However, if **Insulation Specification** is manually defined, select the insulation thickness from those values allowed for the material in the reference data. If the value in the **Insulation Specification** field is a selected reference data insulation specification, the software reads the thickness from the insulation specification from the reference data and displays it here.

Insulation Purpose

Displays the purpose of the insulation. If **Insulation Specification** is set to use the insulation defined by the equipment object, the information in this box cannot be changed. However, if **Insulation Specification** is manually defined, select the purpose of the insulation in this box. Available purposes are read from the reference data.

Insulation Type

Specifies the type of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Material

Specifies the material of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Material** box. However, if **Insulation Specification** is manually defined, you can select the insulation material to use. If **Insulation Specification** is a selected reference data insulation specification, the software reads the material from the insulation specification as defined in the reference data and displays it here.

Select **More** to select an insulation material from the Catalog. To change the options on the list, edit the **Insulation Material** select list in Catalog.

Operating Temperature

Specifies the operating temperature.

Insulation Surface Area

Specifies the measurement of the surface area of insulation.

Heat Tracing Requirement

Indicates whether or not the equipment is heat-traced. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Type

Specifies the type of heat-tracing. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium

Specifies the heat-tracing medium to apply to the object. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium Temperature

Specifies the temperature of the heat-tracing medium. Include the unit of measurement of temperature (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius, for example).

Weight & CG**Dry Weight**

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CGX

Specifies the X-axis location of the dry center-of-gravity.

Dry CGY

Specifies the Y-axis location of the dry center-of-gravity.

Dry CGZ

Specifies the Z-axis location of the dry center-of-gravity.

Wet CGX

Specifies the X-axis location of the wet center-of-gravity.

Wet CGY

Specifies the Y-axis location of the wet center-of-gravity.

Wet CGZ

Specifies the Z-axis location of the wet center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin

Specifies the way in which the wet weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit

the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

See Also

Designed Equipment Properties Dialog Box (on page 44)

Select Insulation Material Dialog Box

Specifies the insulation materials for a selected object. By browsing through the part hierarchy, you can find and select an insulation material in the Catalog database.



This option is unavailable.

Cut

This option is unavailable.



This option is unavailable.



This option is unavailable.

Delete

This option is unavailable.

 **Undo**

This option is unavailable.

 **Insert Row**

This option is unavailable.

 **Move Up**

This option is unavailable.

 **Move Down**

This option is unavailable.

 **Properties**

Displays the properties of the selected object. Because you cannot modify any properties until the object is placed, all properties on the **Properties** dialog box are read-only.

 **Preview**

Displays a bitmap symbol of the selected object. The image file must be assigned to the object in the catalog reference data.

 **Filter**

This option is unavailable.

 **Sort**

This option is unavailable.

 **Customize Current View**

This option is unavailable.

 **List View**

Sets the dialog box to display insulation thickness values in a list view.

 **Grid View**

Sets the dialog box to display insulation thickness values in a spreadsheet-style grid view.

 **Back**

Returns you to the previously selected insulation material folder. Use this command to navigate through the hierarchy to the specific insulation material you need.

 **Forward**

Sends you to the last selected insulation material folder that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific insulation material you need.

 **Up One Level**

Brings up the next highest level of the hierarchy. Use this command to navigate through the hierarchy to the specific material you need.

Check Data

This option is unavailable.

CheckData Ignored Inconsistencies

This option is unavailable.

Smart 3D Help

This option is unavailable.

Address

Sets the current location within the hierarchy of available insulation materials. The dropdown box lists the folders you have visited.

Definition Tab (Equipment Properties Dialog Box)

Displays the part information for the equipment object, the properties and their values, as defined in the reference data. If more than one equipment object is selected, only the common properties for the selected objects display on the tab. For more information about the information defined in the reference data, see the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Category

Select the properties to review. Equipment information is divided into several different categories, such as **Standard**, **Equipment Specification**, **Equipment Support**, **Nozzle Length**, and **Equipment Dimensions**.

 **IMPORTANT** Not all categories are available for all equipment objects.

Property

Displays the name of the property. The properties that appear are dependent on the equipment type selected. For example, the properties displayed for an equipment object based on a pump are different from those based on a cooler. For more information on equipment part properties, see the *Equipment and Furnishings Reference Data Guide*, available with the **Help > Printable Guides** command within the Equipment and Furnishings task.

Value

Displays the value of the corresponding property.

Insulation Tab

Displays insulation properties that have been defined for the selected equipment.

Insulation Specification

Specifies whether or not the selected equipment object is insulated. The options on this dialog box remain disabled unless **User Defined** is selected from the dropdown list.

Purpose

Specifies the insulation purpose for the selected equipment object.

Material

Displays the **Select Insulation Material** dialog box from which you can specify an insulation

material. For more information, see *Select Insulation Material Dialog Box* (on page 34).

Thickness

Defines the thickness of the specified insulation material. This option remains disabled until a material is selected from the **Select Insulation Material** dialog box. By default, the smallest thickness value of the selected material is displayed. An alternative value may be selected from the dropdown list.

See Also

Designed Equipment Properties Dialog Box (on page 44)

Equipment Properties Dialog Box (on page 27)

SECTION 5

Place Equipment Component



Specifies any equipment component from the Equipment Components folder of the catalog, and places an occurrence of that component in the model. You can modify the offset of the component, its relationships to other equipment, and other properties during or after placement. Using positioning relationships, the **Place Equipment Component** command allows you to mate, connect, or align equipment components, and you can use common tools like the **PinPoint** command for precise positioning of the component.

Equipment components represent a sub-portion of an equipment item. For example, you could place a storage tank with support lugs and a platform. The support lugs, tank body, and platform are all considered components of a final functional tank.

Components can be a purchased item that is represented in the catalog as a standard component (such as a tank), or they can be built up of many different shapes and other components (such as an equipment platform). You can place components from a catalog as children of a designed equipment item, but you cannot place them as a child of another equipment component.

You place equipment components into the model by selecting the equipment component part from the **Select Equipment** dialog box and positioning the equipment component in the model using the available relationships.

Place Equipment Component Ribbon

Sets options for adding equipment components to your model. This ribbon appears automatically after you select the **Place Equipment Component** command and then select an equipment object, or when you select an existing equipment component.

TIP To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.

Equipment Properties

Edits the occurrence properties and reviews the static properties of an existing piece of equipment. Equipment properties can be set only after an equipment object is placed in the model. Equipment properties can be edited only after the object is placed in the model.

NOTE Any object modeled in Solid Edge that has occurrence properties cannot be placed or modified unless your computer has a copy of Solid Edge installed.

User Defined Form Definition

Displays **User Defined Form** for the particular equipment if it is defined in the catalog. See the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Relationship List

Lists all relationships for the selected equipment and provides an option for creating a new relationship if the equipment is not already fully constrained. An equipment part is fully constrained when it has sufficient defined relationships to prevent movement or rotation of

the part along all three coordinate axes.

Positioning Relationships

Displays the available options for types of positioning relationships. Some options may not be available for all equipment types. See *Positioning Relationships* (on page 12) for more information.

Delete Relationship

Removes the selected relationship from the equipment model and the database. Using the **Relationships** list box, select a previously existing relationship for the equipment, and click **Delete Relationship**. You can use this command only when modifying existing equipment.

Equipment Component Reference

Prompts you for the reference on the equipment component to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

Second Part Reference

Prompts you for the reference on the equipment object or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first equipment part chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment placement if needed.

Name

Displays the equipment name, as dictated by your predefined name rules, and accepts changes to that name.

Equipment

Specifies the equipment with which to associate the selected equipment component.

Select Equipment Dialog Box

Specifies the equipment needed for placement. This dialog box appears automatically when you click the **Place Equipment**  command. By browsing through the part hierarchy, you can find any piece of equipment in the Catalog database. After you select a piece of equipment and click **OK**, the **Equipment Properties** dialog box appears so that you can define properties for the new equipment. After you click **OK** on the **Properties** dialog box, the software returns you to the model so that you can finalize configuration and placement.

Save

This option is unavailable.

Cut

This option is unavailable.



This option is unavailable.



This option is unavailable.



This option is unavailable.



This option is unavailable.



This option is unavailable.



This option is unavailable.



This option is unavailable.



Displays the equipment properties as defined in the catalog.



Displays a bitmap symbol of the selected equipment. The image file must be assigned to the equipment in the catalog reference data.



This option is unavailable.



This option is unavailable.



This option is unavailable.



Sets the dialog box to display equipment in a list view.



Sets the dialog box to display equipment in a spreadsheet-style grid view.



Returns you to the previously selected equipment part or node. Use this command to navigate through the equipment hierarchy to the specific part you need.



Sends you to the last selected equipment part or node that you moved away from by using the **Back** button. Use this command to navigate through the equipment hierarchy to the

specific part you need.

Up One Level

Brings up the next highest level of the Equipment catalog hierarchy. Use this command to navigate through the equipment hierarchy to the specific part you need.

Check Data

This option is unavailable.

Check Data Ignored Inconsistencies

This option is unavailable.

Help

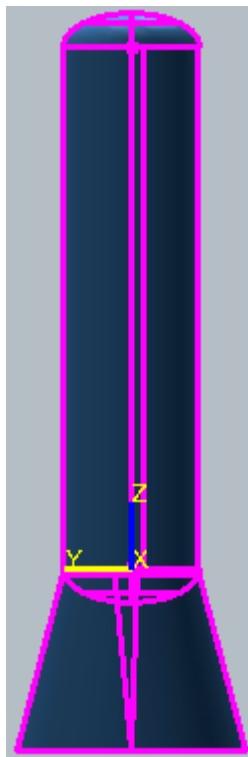
Displays on-line help for Equipment and Furnishings.

Address

Specifies your exact location within the displayed hierarchy.

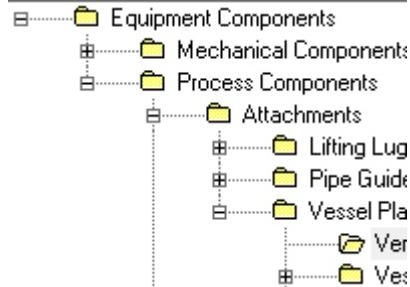
Place equipment component from the catalog

1. Click **Place Equipment Component**  on the vertical toolbar.
2. Select the equipment object to which to add the component in the **Workspace Explorer** or in the model.



The **Select Equipment** dialog box appears.

- Select the equipment component to place from the **Select Equipment Component** dialog box, and click **OK**.

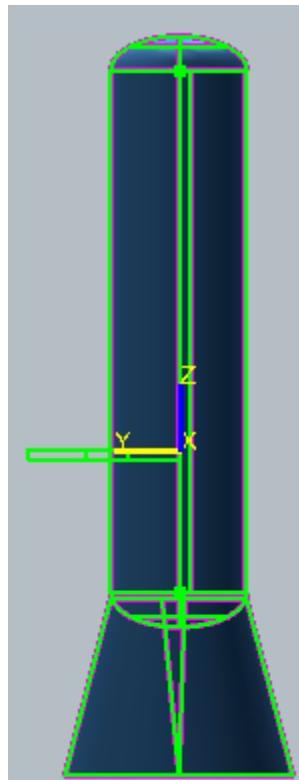


- In the list view, select the component object.

Name	Part Description	Symbol Definition	Definition
VerticalVesselPlat...	VerticalVesselPlat...	SP3DVesselPlat...	SP3DVesselPlat...

The **Equipment Component Properties** dialog box appears.

- Set properties for the new equipment component as needed, and then click **OK**.
- Click in the graphic view to select the mounting surface and approximate position for the equipment component.



TIP Press the left and right arrow keys to rotate the equipment component by 90-degree increments at any time during the placement of the equipment component. You can press

the up arrow key to scroll through the three possible axes of rotation. Press the down arrow key to scroll to each datum point.

7. Using the **Place Equipment Component** ribbon, do any of the following, if needed:
 - Set an offset for the equipment relationship by entering the distance in the **Offset** box.
 - Click **Properties**  on the ribbon, and type any necessary property information in the grid provided on the **Equipment Component Properties** dialog box.

Edit Equipment Properties (on page 26)

TIP You can view the definition properties of the equipment object using the **Properties** command within the **Select Equipment Component** dialog box on the equipment component property page after you place the equipment component. The occurrence properties for an equipment component object can be defined or modified after the equipment component object has been placed in the model.

8. If necessary, add or change a positioning relationship by selecting it from the **Relationship** list on the **Place Equipment Component** ribbon.

Set Positioning Relationships for Equipment (on page 23)

NOTE You cannot place equipment components with occurrence properties modeled using Solid Edge unless Solid Edge is installed on your computer. If you try to place such an object, the following message appears: "Cannot start Solid Edge. Exiting InitializeSymbolDefinition."

IMPORTANT As a workaround to the Solid Edge requirement, you can use the Bulkload utility with the flavors option. Designers can create dozens of variations for any equipment component part imaginable. Also, by creating several variations of a part, rather than using occurrence properties to create the variations automatically, designers can implement custom Solid Edge equipment component without having to install Solid Edge on every designer's computer.

Each variation of a part represents some minor deviation from the catalog part, be it on the basis of size, operational specifications, or material. With Solid Edge, new variations can be created on the basis of size, while the software allows manipulation of the operating parameters or material of construction in the reference data. For more information on bulkloading with flavors, refer to the *Smart 3D Reference Data Guide*, accessible from the **Help > Printable Guides** command in the software.

Equipment Component Properties Dialog Box

Displays equipment component properties for review and editing.

See Also

Occurrence Tab (Equipment Component Properties Dialog Box) (on page 60)

Definition Tab (Equipment Properties Dialog Box) (on page 36)

Connection Tab (on page 37)

Connection Tab (on page 37)

Configuration Tab (on page 148)

Notes Tab (on page 150)

Occurrence Tab (Equipment Component Properties Dialog Box)

Displays all editable instance-specific information about the selected equipment component. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If more than one component is selected, only the common occurrence properties for the selected objects appear on the tab.

When viewing properties for a single component, the following properties appear. More properties may appear depending on what you defined in the reference data. For more information about occurrences defined in the reference data, see the *Equipment and Furnishings Reference Data Guide*, available with the **Help > Printable Guides** command in the software.

NOTE Because equipment component properties are customizable in the equipment and furnishings reference data, only the properties that are required by the software are documented.

Display Equipment Preview

Displays the bitmap image associated with the equipment component if one has been defined in the reference data. The image displays in a separate window.

Category

Select the properties to view, modify, or define. Equipment component properties are divided into several different categories: **Standard, Position and Orientation, Insulation and Tracing, Equipment Dimension, Weight and CG, Fabrication and Construction, Surface Treatment and Coating, and Responsibility**.

Standard

Name

Specifies the name of the object. If a **Name Rule** is specified, then the software uses that rule to determine this name. If the **Name Rule** value is **User Defined**, then you must type a name in this box.

Name Rule

Displays the available name rules for the selected object. Specify the naming rule to use to name the object. You can select one of the listed rules, or you can select **User Defined** to specify the name yourself in the **(Name)** box. **Description**

Specifies a description for the object.

Equipment

Select the equipment to which the component belongs.

Reporting Requirements

Specifies the reporting requirement for the object. To change the options on the list, edit the **Reporting Type** select list in Catalog.

Reporting Type

Specifies the type of reporting. To change the options on the list, edit the **Reporting Type** select list in Catalog.

Behavior Controlled by User

Indicates whether or not you can delete the object. Select **False** to prevent anyone from deleting the object from the model.

Correlation Status

Specifies whether or not the object has been correlated to an object in a P&ID. The list is defined by the EFWCorrelationStatus select list.

Correlation Basis

Specifies whether or not the object is correlated to a P&ID object. The list is defined by the EFWCorrelationBasis select list.

Correlate Object indicates that the object has a correlating object in a P&ID.

No correlation is required indicates that the object does not have a correlating object in a P&ID.

Correlation Approval Status

Specifies whether or not the object is approved with discrepancies in the three-dimensional data compared with design basis data. The list is defined by the EFWCorrelationApprovalStatus sheet in the AllCodelist workbook.

Topology mismatch approved indicates that the object topology mismatch can be ignored.

Data and Topology mismatches approved indicates that the object data and topology mismatches can be ignored.

None indicates that you do not approve a mismatch.

Position and Orientation**East**

Displays the distance of the connection point from the active coordinate system origin in the east direction.

North

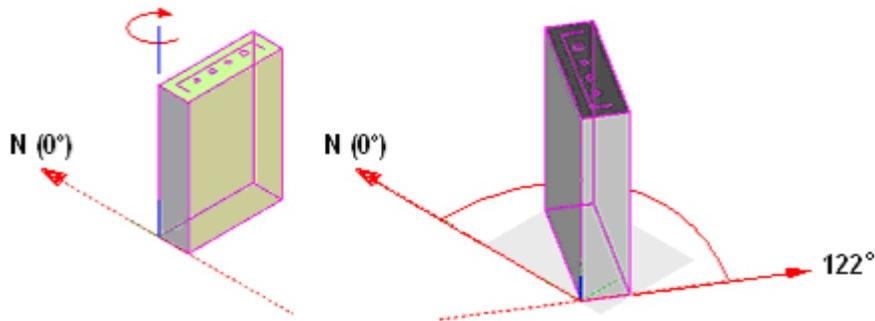
Displays the distance of the connection point from the active coordinate system origin in the north direction.

Elevation

Displays the distance of the connection point above or below the active coordinate system origin.

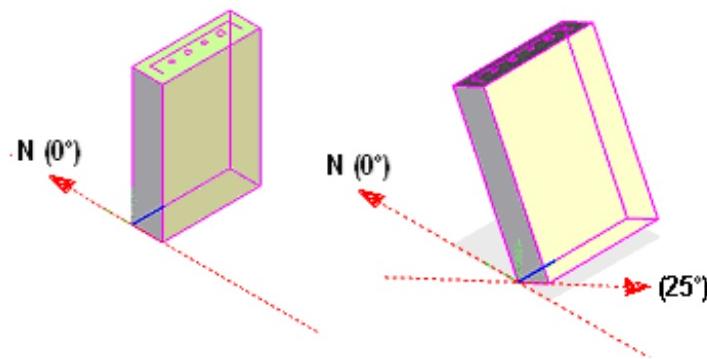
Bearing

Specifies the bearing angle for the object. The **Bearing** angle is measured between the local x-axis of the object and the Y-axis (North) of the global coordinate system in the XY-plane. The local x-axis is the default axis of primary symmetry for all symbols in the catalog. The bearing measurement direction is clockwise from the active coordinate system North looking in the negative active coordinate system direction; that is, down from 0 to 360 degrees. You can enter negative bearing angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the bearing measure displays 0 degrees.



Pitch

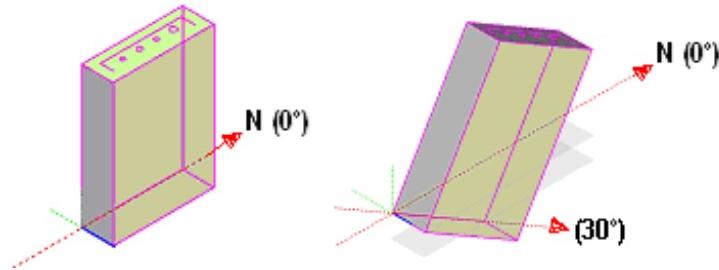
Specifies the pitch angle for the shape. The **Pitch** angle measures between the x-axis of the object and the X-axis of the global coordinate system in the XZ-plane. This option sets the reference in the model to a line that is the intersection of the vertical plane through the X-axis of the local coordinate system and the active coordinate system horizontal plane. The angle is measured in the positive direction from the horizontal plane in the active coordinate system up direction regardless of the current bearing. Another way to describe pitch is the rotation of the object about its y-axis. Pitch angles are limited to between -90 degrees and +90 degrees, with 0 indicating horizontal.



Roll

Specifies the roll angle for the object. The **Roll** angle measures between the local z-axis of the object and the Z-axis of the global coordinate in the YZ-plane. Another way to describe roll is the rotation of the object about its x-axis. This option sets the reference in the model to a line that is perpendicular to the local coordinate system x-axis and in the horizontal plane. Roll angles are measured clockwise from horizontal to the y-axis of the local coordinate system. The roll angle is between 0 and 360 degrees. You can enter negative roll angles, but the software automatically converts them to the positive equivalents. If the pitch is set to

+/- 90 degrees, then the reference in the model is the North axis.



Insulation and Tracing

Is Insulated

Indicates whether or not the object is insulated.

Thickness

Specifies the thickness of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Thickness** box. However, if **Insulation Specification** is manually defined, select the insulation thickness from those values allowed for the material in the reference data. If the value in the **Insulation Specification** field is a selected reference data insulation specification, the software reads the thickness from the insulation specification from the reference data and displays it here.

Requirement

Specifies the requirement of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Insulation Type

Specifies the type of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Insulation Purpose

Specifies the purpose of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Purpose** box. However, if **Insulation Specification** is manually defined, you can select the purpose of the insulation in this box. If **Insulation Specification** is a selected reference data insulation specification, the software reads the purpose from the insulation specification from the reference data and displays it here.

To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Material

Specifies the material of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Material** box. However, if **Insulation Specification** is manually defined, you can select the insulation material to use. If **Insulation Specification** is a selected reference data insulation specification, the software reads the material from the insulation specification as defined in the reference data and displays it here.

Select **More** to select an insulation material from the Catalog. To change the options on the

list, edit the **Insulation Material** select list in Catalog.

Operating Temperature

Specifies the operating temperature.

Insulation Surface Area

Specifies the measurement of the surface area of insulation.

Heat Tracing Requirement

Indicates whether or not the equipment is heat-traced. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Type

Specifies the type of heat-tracing. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium

Specifies the heat-tracing medium to apply to the object. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium Temperature

Specifies the temperature of the heat-tracing medium. Include the unit of measurement of temperature (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius, for example).

Weight & CG

Displays the center-of-gravity and the weight of the selected equipment objects. The center-of-gravity locations are displayed in global system coordinates along the X-, Y-, and Z-axes.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CGX

Specifies the X-axis location of the dry center-of-gravity.

Dry CGY

Specifies the Y-axis location of the dry center-of-gravity.

Dry CGZ

Specifies the Z-axis location of the dry center-of-gravity.

Wet CGX

Specifies the X-axis location of the wet center-of-gravity.

Wet CGY

Specifies the Y-axis location of the wet center-of-gravity.

Wet CGZ

Specifies the Z-axis location of the wet center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin

Specifies the way in which the wet weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction**Fabrication Requirement**

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating**Exterior Coating Requirement**

Select the coating requirement for the object. To add, edit, or remove values that are available for selection, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog. If you want to add, edit, or remove values that are available for selection, edit the **Compart Coating Type** sheet in the CompartmentCodeLists.xls workbook in the reference data. Smart 3D includes this property

in the painting area report.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

Definition Tab (Equipment Properties Dialog Box)

Displays the part information for the equipment object, the properties and their values, as defined in the reference data. If more than one equipment object is selected, only the common properties for the selected objects display on the tab. For more information about the information defined in the reference data, see the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Category

Select the properties to review. Equipment information is divided into several different categories, such as **Standard**, **Equipment Specification**, **Equipment Support**, **Nozzle Length**, and **Equipment Dimensions**.

★ **IMPORTANT** Not all categories are available for all equipment objects.

Property

Displays the name of the property. The properties that appear are dependent on the equipment type selected. For example, the properties displayed for an equipment object based on a pump are different from those based on a cooler. For more information on equipment part properties, see the *Equipment and Furnishings Reference Data Guide*, available with the **Help > Printable Guides** command within the Equipment and Furnishings task.

Value

Displays the value of the corresponding property.

SECTION 6

Place Designed Equipment Component

 Specifies any equipment component from the Equipment Components folder of the catalog and places an occurrence of it inside the model. You can modify the offset of the component, its relationships to other equipment, and other properties during or after placement. Using positioning relationships, the **Place Designed Equipment Component** command allows you to mate, connect, or align equipment components, and you can use common tools like the **PinPoint** command for precise positioning of the component.

The goal of the software's designed equipment component modeling capabilities is to allow you to build an equipment component definition in the Model database by combining basic shapes, ports, such as nozzles, foundations, electrical, and so on, and properties defined in the reference data.

Designed equipment component objects can be built entirely of primitive geometric shapes. For example, you can use the **Place Designed Equipment Component**  command to create a designed equipment component object in the model and add lugs, using the **Place Shape**  command, to customize the designed equipment component.

You can also place a nozzle using a nozzle defined in the P&ID design basis (if P&ID design basis data is available).

Some equipment component shapes can be modeled more easily using solid modeling software. In addition to using the primitive shapes delivered with the software, you can import graphics from an SAT file as the geometry for the designed equipment shape. After the geometry is defined, you can then place ports to define distributed connections to the designed equipment.

Place Designed Equipment Component Ribbon

Sets options for adding designed equipment components to your model. This ribbon appears automatically after you select the **Place Designed Equipment Component**  command and then select an equipment object, or when you select an existing equipment component.

 **TIP** To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.

Equipment Component Properties

Edits the occurrence properties and reviews the static properties of an existing equipment component. Equipment components properties can be set and edited only after an equipment component object is placed in the model.

 **NOTE** Any object modeled in Solid Edge that has occurrence properties cannot be placed or modified unless your computer has a copy of Solid Edge installed.

User Defined Form Definition

Displays **User Defined Form** for the particular equipment if it is defined in the catalog. See the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Relationship List

Lists all relationships for the selected equipment component and provides an option for creating a new relationship if the equipment component is not already fully constrained. An equipment component part is fully constrained when it has sufficient defined relationships to prevent movement or rotation of the part along all three coordinate axes.

Positioning Relationships

Displays the available options for types of positioning relationships. See *Positioning Relationships* (on page 12) for more information.

Delete Relationship

Removes the selected relationship from the equipment component model and the database. Using the **Relationships** list box, select a previously existing relationship for the equipment component, and click **Delete Relationship**. You can use this command only when modifying existing equipment components.

Equipment Component Reference

Prompts you for the reference on the equipment component to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

Second Part Reference

Prompts you for the reference on the equipment component object or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first equipment component part chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment component placement if needed.

Name

Displays the equipment component name, as dictated by your predefined name rules, and accepts changes to that name.

Equipment

Specifies the equipment with which to associate the selected equipment component.

Place designed equipment component

1. Click **Place Designed Equipment Component**  on the vertical toolbar.
2. Select the equipment to which to add the component from the model or the **Workspace Explorer**.

The **Select Equipment Component Type** dialog box appears.

3. Select the equipment component to place, and click **OK**.

The **Design Equipment Component Properties** dialog box appears.

4. Set properties as needed for the new designed equipment component, and click **OK**.

The software returns you to the graphic window so you can define the position of the equipment component.

5. Using the **Place Designed Equipment Component** ribbon, do any of the following, if necessary:

- Set an offset for the equipment relationship by typing the distance in the **Offset** box.
- Click **Properties**  on the ribbon, and type any necessary property information in the grid provided on the **Equipment Component Properties** dialog box.

Edit Equipment Properties (on page 26)

TIP You can view the definition properties of the equipment object using the **Properties** command within the **Select Equipment Component** dialog box on the equipment component property page after you place the equipment component. The occurrence properties for an equipment component object can be defined or modified after the equipment component object has been placed in the model.

6. If necessary, add or change a positioning relationship by selecting it from the **Relationship** dropdown list on the **Place Equipment Component** ribbon.

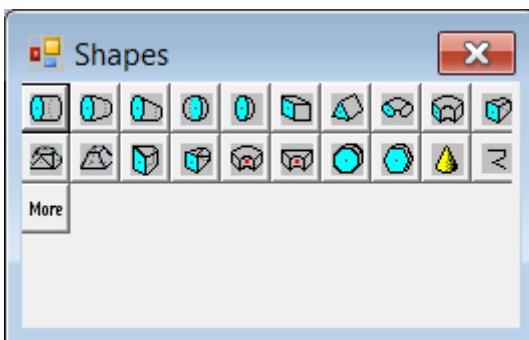
Set Positioning Relationships for Equipment (on page 23)

SECTION 7

Place Shape

 Adds additional shapes to an existing equipment or equipment component type. The **Place Shape** command uses the equipment or equipment component object selected in the **Workspace Explorer** hierarchy. If no equipment or equipment component object is selected prior to starting the command, you are prompted to select an equipment or equipment component object.

If you click and hold the button on the vertical toolbar, the **Place Shape** palette displays so that you can select a different type of shape to place.



Click **More** to display the **Select Shape** dialog box and select a shape from those available in the Catalog database.

Place Shape Ribbon

Sets options for positioning the specific shape as part of a selected designed equipment object.

Properties

Edits the occurrence properties and reviews the static properties of an existing shape. Properties can be edited only after the object is placed in the model. For more information, see *Shape Properties Dialog Box* (on page 76).

Relationship List

Shows the relationship currently applied to the shape and allows you to change or add new relationship as needed.

Positioning Relationship

Displays the available options for types of positioning relationships: Mate, Align, and Connect. Some options may not be available for all shape types. For more information, see *Positioning Relationships* (on page 12).

Delete Relationship

Removes the selected relationship from the model. Using the **Relationships List** box, select a previously existing relationship for the shape, and click **Delete Relationship**. You can use this command only when modifying existing shapes. This control is available only after a shape has been added to the equipment or equipment component.

 **Shape Reference**

Prompts you for the reference on the shape to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

 **Second Part Reference**

Prompts you for the reference on the shape object already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first shape chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment placement if needed. This control is only available after a shape has been added to the designed equipment.

Name

Displays the shape name, as dictated by your predefined name rules, and accepts changes to that name.

 **NOTES**

- If the name of the parent equipment changes, the software automatically updates the name of the shape to reflect that change. For example, if the name of the parent item changes from Pump01 to Pump100, the shape's name changes from Pump01-Shape-001 to Pump100-Shape-001.
- Some properties for the shape may be read-only on the **Place Shape** ribbon, depending on the type of shape you selected.

Equipment

Specifies the equipment or equipment component object with which to associate the selected shape.

Add Shape/Subtract Shape/Suppress Shape

Defines how the shape is used for a designed solid. This option is only available when you are placing or editing a shape in a parent designed solid.

 **Add Shape**

Select to have the shape add its material to the parent designed solid.

 **Subtract Shape**

Select to have the shape subtract its material from the parent designed solid.

 **Suppress Shape**

Select to have the shape added to the parent designed solid in the hierarchy, but have the shape's material ignored--neither added to nor subtracted from the parent designed solid. Use this option to temporarily remove a shape's effect on the parent designed solid, but not remove the shape from the designed solid entirely. You can use suppressed shapes as construction graphics for the placement of other shapes in the designed solid.

Select Shape Dialog Box

Specifies a shape to place on an equipment or equipment object. The dialog box appears automatically when you click **More** in the **Shape** floating palette. By browsing through the shapes hierarchy, you can find any shape that exists in the Catalog database. After you select a shape, the **Shape Properties** dialog box appears so you can define shape properties. When you click **OK** on the **Shape Properties** dialog box, the software returns you to the graphic window so you can place the shape.

Back

Returns you to the previously selected shape type or node. Use this command to navigate through the hierarchy to the specific shape you need.

Forward

Sends you to the last selected shape type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific shape you need.

Up One Level

Brings up the next highest level of the Shapes catalog hierarchy. Use this command to navigate through the hierarchy to the specific shape you need.

Properties

Displays the properties of the selected shape. Because you cannot modify any properties until the shape is placed, all properties on the **Properties** dialog box are read-only.

Preview

Displays a bitmap symbol of the selected shape. The image file must be assigned to the shape in the catalog reference data.

List View

Sets the dialog box to display shapes in a list view.

Grid View

Sets the dialog box to display shapes in a spreadsheet-style grid view.

Address

Specifies your exact location within the displayed hierarchy.

What do you want to do?

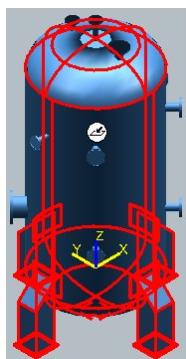
- *Place a shape* (on page 74)
 - *Edit shape properties* (on page 75)
 - *Edit prismatic shape properties* (on page 76)
-

Place a shape

1. Click **Place Shape**  on the vertical toolbar, and hold down a few seconds to display the floating **Shapes** palette.

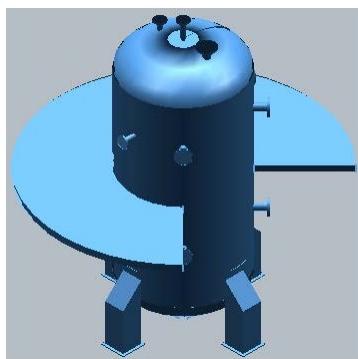


2. Select the shape to place from the **Shapes** palette. The icon of the last used shape displays on the toolbar.



TIP If an equipment or an equipment component object has not been selected, you are prompted to select one. Do this either in a graphic view or in the system hierarchy in the **Workspace Explorer**.

3. If necessary, make adjustments on the **Shape Properties** dialog box, and click **OK**.
4. Click in the graphic view to select an approximate location or reference element for a relationship for the shape.



■ NOTES

- You can continue using the **Place Shape**  command to combine geometric shapes and create customized equipment or equipment component objects.
- After a shape has been placed in the model as part of the equipment or equipment component object, you can use the horizontal ribbon to add or change the positioning relationship.
- Press the left or right arrow keys to rotate the shape by 90-degree increments at any time during the placement of the shape. Press the up arrow to scroll through the three possible axes of rotation. Press the down arrow key to scroll to each datum point.

Edit shape properties

1. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a shape in the graphic window.
! TIP If the **Locate Filter** box is not displayed on the horizontal ribbon, click **Select Tool**  on the vertical toolbar.
2. Click **Shape Properties**  on the horizontal ribbon.
! TIP You can also access the **Shape Properties** dialog box by selecting **Properties** from the **Edit** menu, or by right-clicking the shape and choosing **Properties** from the short-cut menu.
3. On the *Occurrence Tab (Shape Properties Dialog Box)* (on page 77), modify the geometric dimensions of the shape.
4. Access the other dialog box tabs, and modify the properties as needed.
5. Click **OK** to save your changes and return to the workspace.

! TIP You can also select **Apply** to put the changes into effect and continue working in the **Properties** dialog box.

■ NOTES

- Each geometric shape object has its own **Properties** dialog box that displays its corresponding parameters, including any reference graphics that illustrate what the dimensional parameters represent.
- All occurrence and definition properties for an equipment object are defined by the Excel workbook named **Equipment.xls** in the reference data. For information on adding or editing different types of properties, refer to the *Reference Data Guide*, accessible from the **Help > Printable Guides** command.

Edit prismatic shape properties

1. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.

TIP If the **Locate Filter** box is not displayed on the horizontal ribbon, click the **Select Tool**  on the vertical toolbar.
2. Click **Properties**  on the horizontal ribbon.

TIP You can also access the **Prismatic Shape Properties** dialog box from the **Relationship** tab on the **Designed Equipment Properties** dialog box. Select the name of the prismatic shape to edit, and click **Go To**.
3. On the **Occurrence** tab, modify the display aspect or the name of the prismatic shape.
4. On the **Cross-Section** tab, modify the geometric dimensions of the existing cross-section, or select a new cross-section and type new property information.
5. Access the other dialog box tabs, and modify the properties as needed.
6. Click **OK** to save your changes and return to the workspace.

TIP You can also select **Apply** to put the changes into effect and continue working in the **Properties** dialog box.

NOTES

- Each geometric shape object has its own **Properties** dialog box that displays its corresponding parameters, including any reference graphics that illustrate what the dimensional parameters represent.
- All occurrence and definition properties for an equipment object are defined by the Excel workbook named **Equipment.xls** in the reference data. For information on adding or editing different types of properties, refer to the *Reference Data Guide*, accessible from the **Help > Printable Guides** command in the software.

Shape Properties Dialog Box

Displays shape properties for review and editing.

See Also

- Occurrence Tab (Shape Properties Dialog Box) (on page 77)*
Configuration Tab (on page 148)
Notes Tab (on page 150)

Occurrence Tab (Shape Properties Dialog Box)

Sets properties for the unique instance of the selected shape. Shapes can be placed either from what has been defined in the reference data, or by importing a geometric shape contained within an SAT file.

Category

Displays the defined category name for the selected designed equipment. This data is retrieved from the Equipment and Furnishings reference data.

Display Aspect

Displays the **Display Aspect** dialog box which lists all the display aspects that may be applied to the selected shape.

Property

Lists all the dimensional properties of the selected shape. You can add or modify these properties through the catalog reference data. The properties available change depending on the type of shape selected.

Value

Shows the current values for all properties of the selected shape. You can modify these values to reflect exact design needs or deviations from the standard part.

Name

Specifies the name of the object. If a **Name Rule** is specified, then the software uses that rule to determine this name. If the **Name Rule** value is **User Defined**, then you must type a name in this box.

Name Rule

Displays the available name rules for the selected object. Specify the naming rule to use to name the object. You can select one of the listed rules, or you can select **User Defined** to specify the name yourself in the **(Name)** box. **Behavior Controlled by User**

Indicates whether or not you can delete the object. Select **False** to prevent anyone from deleting the object from the model.

Prismatic Shape Properties Dialog Box

Displays prismatic shape properties for review and editing.

See Also

Occurrence Tab (Prismatic Shape Properties Dialog Box) (on page 78)

Cross-Section Tab (Prismatic Shape Properties Dialog Box) (on page 78)

Configuration Tab (on page 148)

Notes Tab (on page 150)

Occurrence Tab (Prismatic Shape Properties Dialog Box)

Sets properties for the unique instance of the selected prismatic shape. Prismatic shapes can be placed in the model by selecting **PrismaticShape** in the **Shapes** dropdown list on the *Occurrence Tab (Shape Properties Dialog Box)* (on page 77).

Category

Displays the defined category name for the selected prismatic shape. This data is retrieved from the Equipment and Furnishings reference data.

Shape

Displays the **PrismaticShape** icon. This option is read-only when you access the tab to edit a prismatic shape that has already been placed in the model.

Display Aspect

Lists all the display aspects that can be applied to the selected shape.

Name

Displays the name defined for the selected prismatic shape during its initial placement in the model.

Name Rule

Displays the available name rules for the selected object. Specify the naming rule to use to name the object. You can select one of the listed rules, or you can select **User Defined** to specify the name yourself in the **(Name)** box.

Cross-Section Tab (Prismatic Shape Properties Dialog Box)

Displays information about the cross-section for a prismatic shape that was projected along a path.

Cross Section

Specifies the type of cross-section for the object. You can either select a standard cross-section defined in the reference data or select **Sketch** to draw your own cross-section.

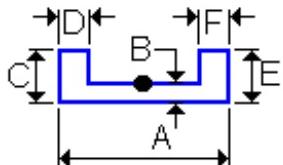
Select any standard cross-section type in the **Cross Section** list to modify the properties that are described in this topic. When you select **Sketch** in the **Cross Section** list, you can view options, such as each point, its X-, Y-, and Z-coordinates in the model, and its turn type, but you cannot modify them.

For more information, see *General Tab (Sketch Properties Dialog Box)* (on page 105).

Display Cross Section Image

Displays the image associated with the standard cross-section type in the reference data.

Cross-section images show the dimensions that you can define for the cross-section type, the default cardinal points for the cross-section type, and the angle for the cross-section. For example, the following graphic contains the dimensions and cardinal point for a standard road cross-section:



If no image is associated with the cross-section and the selected cardinal point, **Image Not Available** displays at the bottom of the tab.

A - G

Defines the dimensions for standard cross-sections. If an image is defined for the cross-section in the reference data, you can see what each letter represents by clicking **Display Cross Section Image** .

Cardinality

Defines the point where the software attaches the cross-section to the path. If you click **Display Cross-Section Image**  to see a graphic of the cross-section, you can view where each cardinal point is located by selecting each cardinal point in the list. The software automatically updates the display, if the appropriate graphic is available, with a graphic containing the selected cardinal point.

You can select **User Defined** cardinal points to specify a reference point on selected cross section for precise placement.

Angle

Defines the angle by which the cross-section is rotated about the path.

SECTION 8

Creating Customized Shapes

In the Equipment and Furnishings task, you can create customized shapes in the location that you need using the place prismatic shape feature. This feature is especially useful for designed equipment objects that need to have an unusual shape.

When you place a prismatic shape, a two-dimensional cross-section is projected along a path that you specify in the model to create the shape. The path determines the actual location of the shape in the model. The cross-section, on the other hand, defines the shape and its dimensions.

Defining Paths

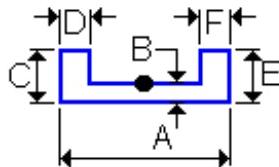
When you define the path along which the cross-section will be projected, you can choose from straight lines or arcs. You can also control all aspects of the path by specifying the types of turns that you need, the dimensions of the turns, and the plane for the path.

Defining Cross-sections

When you define the cross-section for the shape, you can select from a standard set of cross-sections, defined in the reference data, or you can sketch your own two-dimensional cross-section.

When you use a standard cross-sectional type, you can change the dimensions and the cardinal point of the cross-section on the **Cross-Section** tab of the **Shape Properties** or **Prismatic Shape Properties** dialog box. The cardinal point of the cross-section is important in determining the shape and location of the resulting volume. The cardinal point is the point where the software attaches the cross-section to the path. All standard cross-section types have various cardinal points from which you can choose.

For example, in the following graphic, you can see that the cardinal point is located in the center of the road cross-section. If the cardinal point were moved to another location, the actual path of the roadway would be different.



When you sketch a cross-section, you must sketch the cross-section on the two-dimensional plane that is orthogonal to the first leg of the path. The software displays this plane, which is perpendicular to the path, as you sketch the cross-section. The cardinal point is defined as you sketch. In other words, the cross-section surrounds the path and is attached to the path exactly as you sketch it.

Place Prismatic Shape Ribbon

Sets options for placing shapes defined by a path and the cross-section projected along the path.

 **Shape Properties**

Displays the **Prismatic Shape Properties** dialog box, on which you can set properties for the shape that you need to place. For more information, see Prismatic Shape Properties Dialog Box.

 **Path**

Displays the **Create Path** ribbon, which defines the path along which the cross-section is projected.

 **Cross-Section**

Sets properties for the cross-section. If one of the standard cross-section types is selected in the **Cross-Section Type** box, this button is unavailable. If you select **Sketch** as the cross-section type, this button displays the **Create Path** ribbon to allow you to sketch the two-dimensional cross-section. The **Cross-Section** button is only available after you define a path for the volume or when you select **User Defined** for **Cardinality**.

Finish

Places the shape along the path specified with the specified cross-section.

Cross-Section Type

Specifies the type of cross-section to project along the specified path. You can either select a standard cross-section defined in the reference data, or you can sketch your own custom cross-section.

Relationship List

Lists all relationships for the selected shape and provides an option for creating a new relationship if the object is not already fully constrained. An object is fully constrained when it has sufficiently defined relationships to prevent movement or rotation of the object along all three coordinate axes.

Positioning Relationships

Displays the available options for types of positioning relationships: Mate, Align, and Connect. Some options may not be available for all shape types. See *Positioning Relationships* (on page 12) for more information.

 **Delete Relationship**

Removes the selected relationship from the model. Using the **Relationships List** box, select a previously existing relationship for the shape, and click **Delete Relationship**. You can use this command only when modifying existing designed equipment. This control is available only after a shape has been added to the designed equipment.

 **Shape Reference**

Prompts you for the face of the shape to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed. This control is available only after a shape has been added to the designed equipment.

 **Second Part Reference**

Prompts you for the reference on the shape object already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first shape part chosen with respect to the second part selected in the definition of the

relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial placement if needed. This control is available only after a shape has been added to the designed equipment.

Name

Displays the shape name, as dictated by your predefined name rules, and accepts changes to that name.

Equipment

Specifies the designed equipment object with which to associate the selected shape.

Create Path Ribbon

Sets options for defining a new path.

Sketch Properties

Displays the **Sketch Properties** dialog box (on page 105), in which you can view properties for the path.

Finish Path

Displays the path in the active view and returns to the model with the place prismatic shape feature still enabled.

Cancel

Cancels the changes you made and returns you to the model.

Edit

Modifies and moves the existing path. When you initially create a path, this option is available only after you place at least two points in the path. You can select the segment, turn, or multiple segments to which to make modifications.

Create

Sketches the path or adds segments to an existing path.

Origin

Defines the origin on the cross-section for prismatic shape construction.

Reference Point

Specifies that you are currently defining the first point of the path segment.

End Point (Straight Line)

Specifies that you are currently defining the second point of a straight path segment.

End Point (Arc)

Specifies that you are currently defining the second point of an arc. This option appears only when **Arc** is selected in the **Line Type** list.

 **Third Point (Arc)**

Specifies that you are defining the final point of an arc.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Line Type** list.

Line Type Options **Line**

Defines the line type for the segment to be a straight line.

 **Arc by 3 Points**

Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.

 **No Line**

Specifies that you do not want the current segment of the path to have a line associated with it.

Plane

Activates options for selecting a working plane for the path.

Working Plane Options **Plan Plane**

Defines the work surface as the XY plane.

 **Elevation Plane: East-West**

Defines the work surface as the XZ plane.

 **Section Plane: North-South**

Defines the work surface as the YZ plane.

 **Plane by Turn**

Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane.

 **Plane by Three Points**

Defines the work surface using three points that you define.

 **No Plane**

Clears any work surfaces. The software does not project points that you place to any plane.

 **Lock Angle**

Locks or unlocks the **Angle** box. Locking the corresponding angle value creates a constraint along which the selected turn angle can be moved.

Angle

Specifies the angle for the turn.

Lock Length

Locks or unlocks the **Length** box.

Length

Specifies the length of the selected path segment.

Turn Type

Specifies the type of turn associated with the current path segment. The **Turn Type** option is unavailable if you select **Arc** or **No Line** in the **Line Type** list.

Turn Type Options

None

Indicates that no special turn type will be applied to the turn.

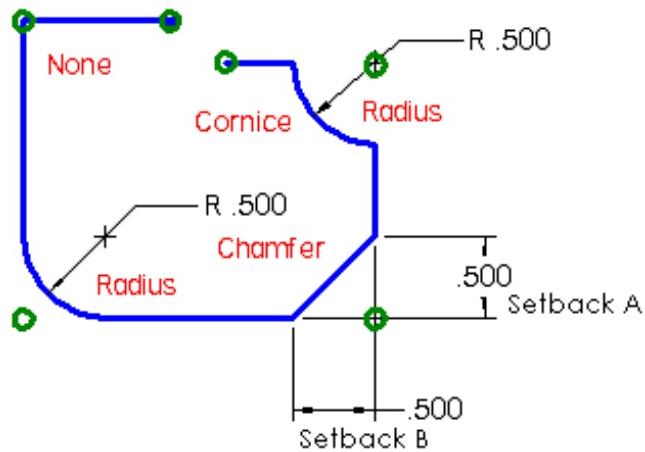
Bend

Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Feature Value** box.

Cornice

Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Feature Value** box.

The following graphic includes an example of each of the available turn types:



Turn Type Value

Defines dimensions for the selected turn type.

Edit Path Straight Segment Ribbon

Sets options for modifying a straight segment of an existing path.

Sketch Properties

Displays the *Sketch Properties dialog box* (on page 105), on which you can view properties for the path.

Finish Path

Displays the path in the active view and returns to the model with the place prismatic shape feature still enabled.

Cancel

Cancels the changes you made and returns you to the model.

Edit

Modifies and moves the existing path. You can select the segment, turn, or multiple segments to which to make modifications.

Create

Displays the **Create Path** ribbon to allow you to add segments to an existing path.

Origin

Defines the origin on the cross-section for prismatic shape construction.

Reference Point

Specifies that you are selecting the starting location of the move vector.

End Point

Specifies that you are selecting the ending location of the move vector.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Line Type** list.

Path Type Options **Line**

Defines the line type for the segment to be a straight line.

Arc by 3 Points

Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.

No Line

Specifies that you do not want the current segment of the path to have a line associated with it.

Plane

Activates options for selecting a working plane for the path.

Working Plane Options **Plan Plane**

Defines the work surface as the XY plane.

Elevation Plane: East-West

Defines the work surface as the XZ plane.

 **Section Plane: North-South**

Defines the work surface as the YZ plane.

 **Plane by Turn**

Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane.

 **Plane by Three Points**

Defines the work surface using three points that you define.

 **No Plane**

Clears any work surfaces. The software does not project points that you place to any plane.

 **Delete Selected Items**

Deletes the selected path segments.

 **Length Locked**

Defines whether or not the length of the selected segment should remain constant while moving.

When locked , the software automatically modifies the turn points, along with the length and angle of adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.

When not locked , the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.

Edit Path Arc Ribbon

Sets options for modifying an arc that is part of an existing path.

 **Sketch Properties**

Displays the *Sketch Properties dialog box* (on page 105), on which you can view properties for the path.

Finish Path

Displays the path in the active view and returns to the model with the place prismatic shape feature still enabled.

Cancel

Cancels the changes you made and returns you to the model.

Edit

Modifies and moves the existing path. You can select the segment, turn, or multiple segments to which to make modifications.

Create

Displays the **Create Path** ribbon to allow you to add segments to an existing path.

Origin

Defines the origin on the cross-section for prismatic shape construction.

 **Reference Point**

Specifies that you are selecting the starting location of the move vector.

 **End Point**

Specifies that you are selecting the ending location of the move vector.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Line Type** list.

Path Type Options **Line**

Defines the line type for the segment to be a straight line.

 **Arc by 3 Points**

Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.

 **No Line**

Specifies that you do not want the current segment of the path to have a line associated with it.

Working Plane Options **Plan Plane**

Defines the work surface as the XY plane.

 **Elevation Plane: East-West**

Defines the work surface as the XZ plane.

 **Section Plane: North-South**

Defines the work surface as the YZ plane.

 **Plane by Turn**

Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane.

 **Plane by Three Points**

Defines the work surface using three points that you define.

 **No Plane**

Clears any work surfaces. The software does not project points that you place to any plane.

 **Delete Selected Items**

Deletes the selected path segments.

Edit Path Turn Ribbon

Sets options for modifying a turn in an existing path.

 **Sketch Properties**

Displays the *Sketch Properties dialog box* (on page 105), on which you can view properties for the path.

Finish Path

Displays the path in the active view and returns to the model with the place prismatic shape feature still enabled.

Cancel

Cancels the changes you made and returns you to the model.

Edit

Modifies and moves the existing path. You can select the segment, turn, or multiple segments to which to make modifications.

Create

Displays the Create Path Ribbon to allow you to add segments to an existing path.

Origin

Defines the origin on the cross-section for prismatic shape construction.

 **Reference Point**

Specifies that you are selecting the starting location of the move vector.

 **End Point**

Specifies that you are selecting the ending location of the move vector.

Plane

Activates options for selecting a working plane for the path.

Working Plane Options **Plan Plane**

Defines the work surface as the XY plane.

 **Elevation Plane: East-West**

Defines the work surface as the XZ plane.

 **Section Plane: North-South**

Defines the work surface as the YZ plane.

 **Plane by Turn**

Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane.

 **Plane by Three Points**

Defines the work surface using three points that you define.

 **No Plane**

Clears any work surfaces. The software does not project points that you place to any plane.

Delete Selected Items

Deletes the selected path segments.

Angle 1

Displays the first angle used in the turn, if one exists. This option is read-only.

Angle 2

Displays the second angle used in the turn, if one exists. This option is read-only.

Angle 3

Displays the third angle used in the turn, if one exists. This option is read-only.

Turn Type

Specifies the type of turn. You can change the turn type by selecting another type in the list.

Turn Type Options

None

Indicates that no special turn type will be applied to the turn.

Bend

Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Feature Value** box.

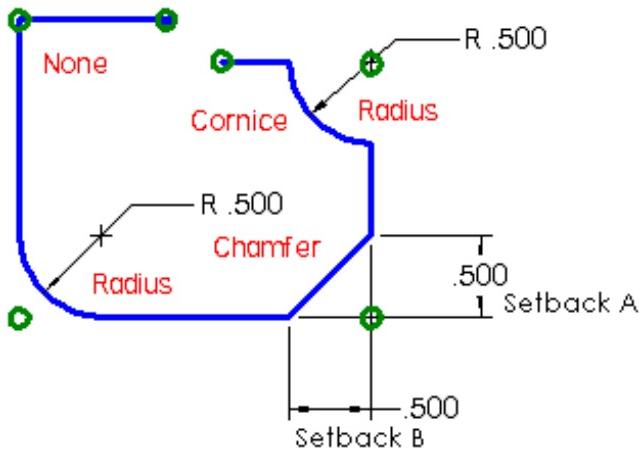
Cornice

Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Feature Value** box.

Chamfer

Specifies that the turn type between two segments is a chamfer. You can specify the dimensions for setback A and setback B of the chamfer in the **Feature Value** box. The dimensions of setback A and setback B for the chamfer must be the same.

The following graphic includes an example of each of the available turn types:



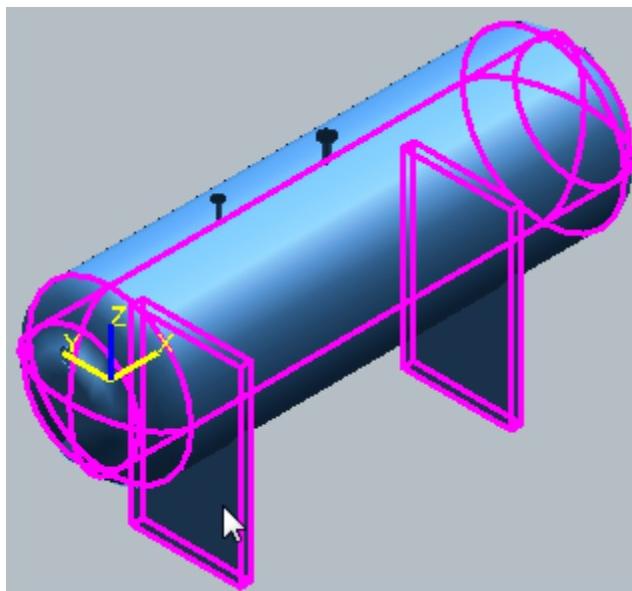
Turn Type Value

Defines dimensions for the selected turn type.

Place a prismatic shape

1. Click **Place Shape**  on the vertical toolbar.

TIP If a designed equipment object has not been selected, select one in a graphic view or in the **Workspace Explorer**.

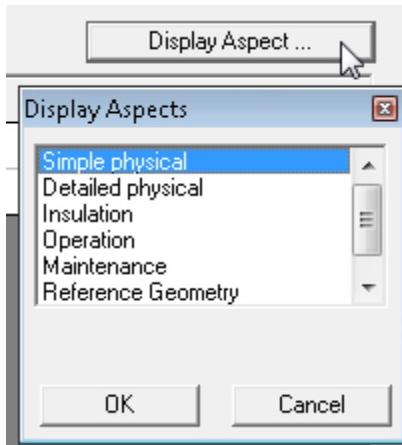


The **Shapes** palette displays

2. On the **Shapes** dialog box, select  **Prismatic Shape**.

TIP Click **More** to select a shape from the catalog browser.

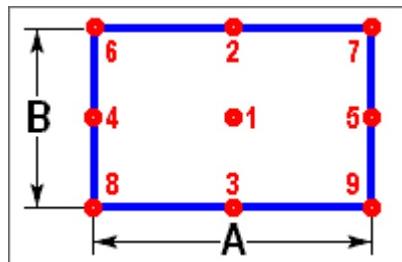
3. Click **Display Aspect** and select the aspects from the list.



TIP You can click **Format > View** and change the **Render Selected Aspects** option in the **Format View** dialog box to display the aspect in the active graphic view.

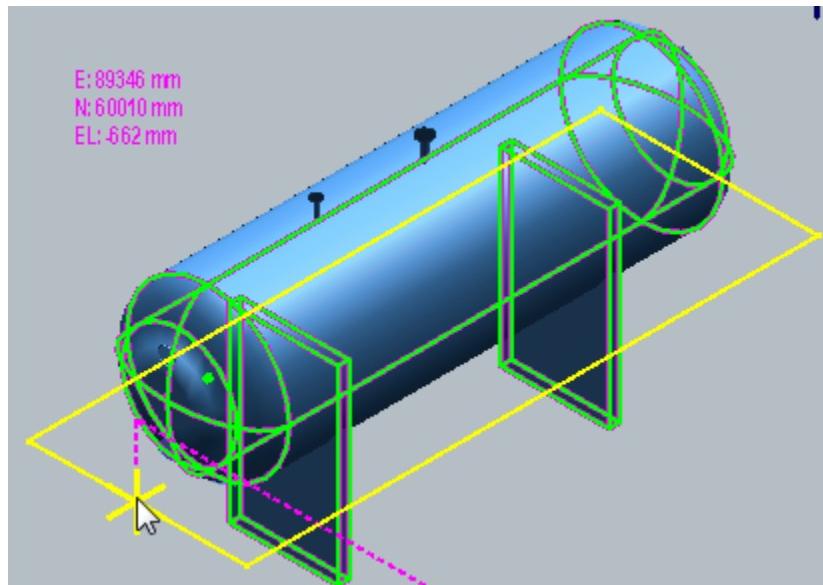
4. Specify a name for the new shape in the **Name** box.

5. On the **Cross-Section** tab, select a cross-section in the drop-down list.



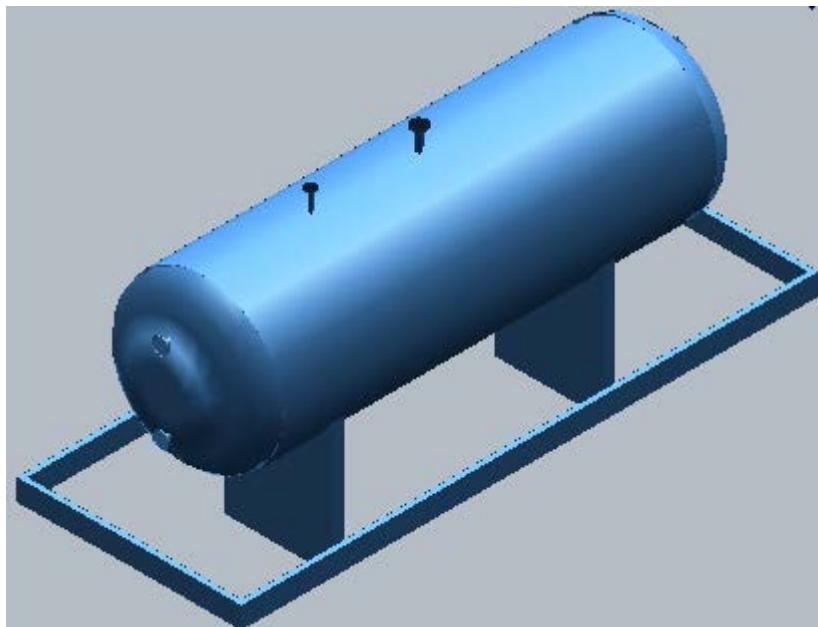
A corresponding image of the selected cross-section appears at the bottom of the dialog box.

6. In the **Value** column, type the appropriate geometric dimensions, cardinal point, and angle.
7. Click **OK** to return to the model.
8. Define the path for the shape. For more information, see *Define the Path for a Prismatic Shape* (on page 92).



9. Click **Finish Path** to finish the path.

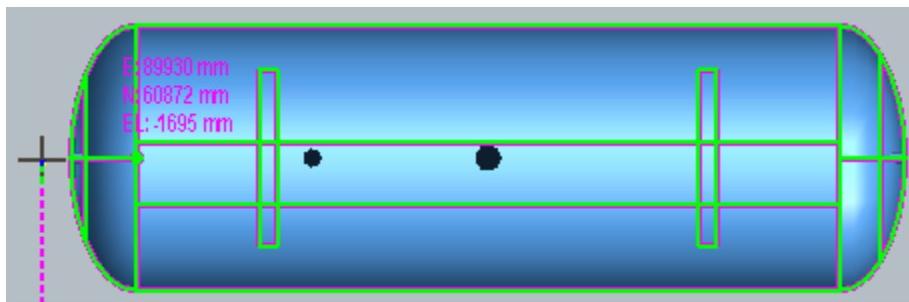
10. Click **Finish** to place the new shape and save it to the database.



11. If necessary, add or change a positioning relationship by selecting it from the **Positioning Relationships** list.

Define the path for a prismatic shape

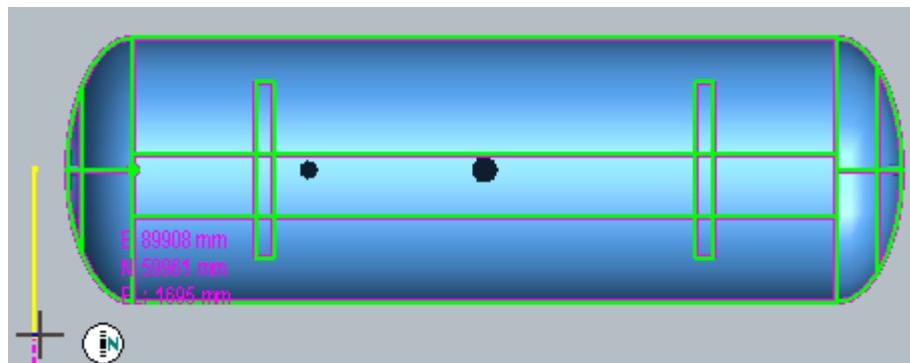
1. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.
- TIP** If the **Locate Filter** box is not displayed on the horizontal ribbon, click the **Select Tool** on the vertical toolbar.
2. On the **Place Prismatic Shape** ribbon, click **Path** .
 3. Click **Create** on the ribbon.
 4. Click the first point for the path.



■ TIPS

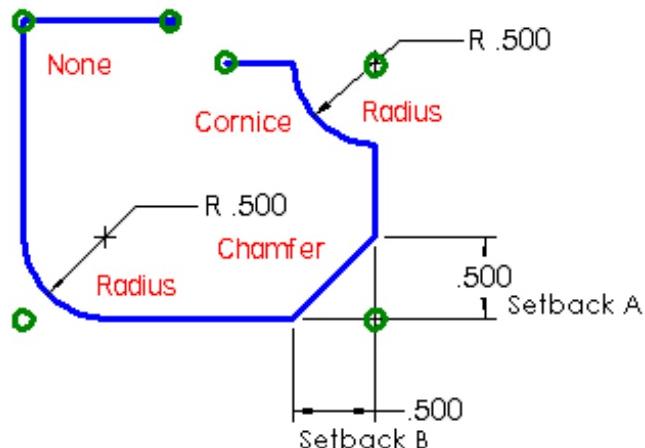
- You can use the **PinPoint** or **Point Along** commands and the SmartSketch relationship indicators when defining your path.

- You can change the plane for the path at any time by selecting the plane in the **Plane** list.
 - To change the segment from a straight line to an arc, click **Arc by 3 Points**  in the **Path Type** list, and then click three points to define the arc.
 - To switch back to a straight line after sketching an arc, click **Line**  in the **Path Type** list.
 - To break the path, click **No Line**  in the **Path Type** list.
5. Click the second point for the path.

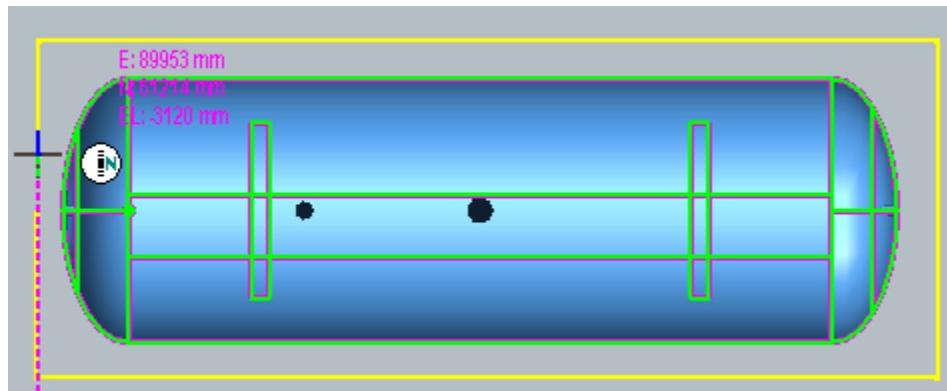


 **TIP** To change the turn type for the corner, click the new turn type in the **Turn Type** list, and define the angle or dimensions for the turn in the **Turn Type Value** box.

The following graphic shows examples of the available turn types:



- Click to place other segments of the path as needed.



- After you place all the points that define the path, click **Finish Path**.

NOTES

- The software does not require that you close the path. When creating a continuous path, you can end it at any point.
- After you place a segment of the path by defining two points, you can click **Edit** on the ribbon to change the segment.

Sketch the cross-section for a prismatic shape

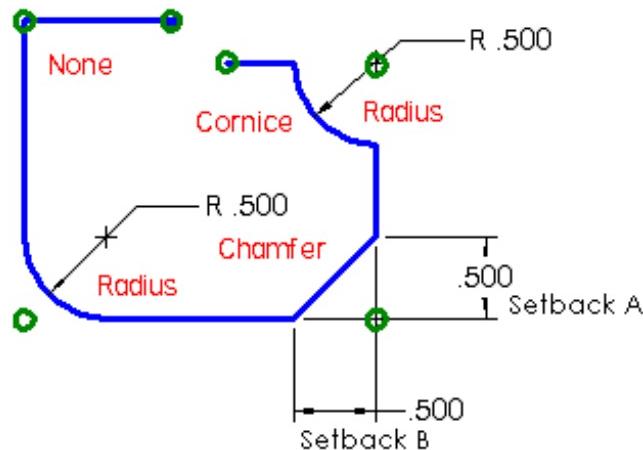
- On the **Place Prismatic Shape** ribbon, select **Sketch** in the **Cross-Section** list.
- Click **Cross-Section** □.
- Click the first point for the cross-section.

TIPS

- You must sketch the cross-section on the two-dimensional plane that is orthogonal to the first leg of the path. Click **Show Cross Section** □ to display a window in the two-dimensional plane. The software displays this plane, which is perpendicular to the path, as you sketch the cross-section.
- The cardinal point, which is the point where the cross-section is attached to the path, is defined as you sketch. In other words, the cross-section surrounds the path and is attached to the path exactly as you sketch it.
- To change the segment from a straight line to an arc, click **Arc by 3 Points** ↗ in the **Path Type** list, and then click three points to define the arc.
- To switch back to a straight line after sketching an arc, click **Line** ✓ in the **Path Type** list.

- Click the next point for the cross-section.

TIP To change the turn type for the corner, click the new turn type in the **Turn Type** list and define the angle or dimensions for the turn in the **Turn Type Value** box. The following graphic shows examples of the available turn types:



5. Click to place other segments of the cross-section as needed.
6. Click the starting point of the cross-section to close it.
7. Click **Finish Path**.

TIP The **Finish Path** button is not available until you close the cross-section by clicking the starting point.

NOTE After you place a segment of the cross-section by defining two points, you can click **Edit** on the ribbon to change the segment.

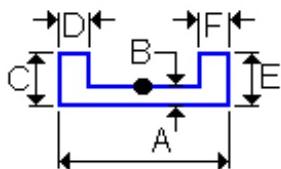
Define cross-sectional properties for a prismatic shape

1. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.
- TIP** If the **Locate Filter** box is not displayed on the horizontal ribbon, click **Select Tool**  on the vertical toolbar.
2. On the **Place Prismatic Shape** ribbon, select a cross-section type in the **Cross- Section Type** list.
- TIP** If you select **Sketch** in the **Cross-Section Type** list, you must sketch the cross-section using the **Path** commands before you can edit properties.
3. Click **Properties** .
4. On the **Prismatic Shape Properties** dialog box, click the **Cross-Section** tab.

- Define values for each dimension of the cross-section. Letters such as A, B, and C represent the cross-sectional dimensions.

!TIPS

- To see what each lettered dimension represents on the cross-section, click **Display Cross-Section Image**  to view a graphic of the selected cross-sectional type if one exists in the reference data.
- Cross-sectional images show the dimensions that you can define for the cross-section type, the default cardinal points for the cross-section type, and the angle for the cross-section. For example, the following graphic contains the dimensions and cardinal point for a standard road cross-section.



Not all cross-section types have images associated with them.

- Select the cardinal point for the cross-section in the **Cardinality** list.

!TIPS

- The cardinal point is the point where the software attaches the cross-section to the path.
- If you click **Display Cross- Section Image**  to see a graphic of the cross-section, you can view where each cardinal point is located by selecting each cardinal point in the list.

- Type an **Angle** for the cross-section, if needed.

- Click **OK**.

■NOTES

- If you sketch a cross-section, the properties for the cross-section include the X-, Y-, and Z-coordinates, the turn type, and values for each point that define the cross-section path.
- You cannot modify the properties of a sketched cross-section on the **Prismatic Shape Properties** dialog box. Instead, you must make changes to the path that defines the cross-section sketch. For more information, see *Modify a Sketched Cross-Section* (on page 96).

Modify a sketched cross-section

- Click **Select** .
- On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape with a sketched cross-section.
- On the ribbon, click **Cross-Section** . The software outlines the cross-section path in yellow.
- To modify a straight segment in the cross-section, select the segment to modify, and then make changes on the ribbon.

Modify a Straight Segment in a Path (on page 101)

5. To modify an arc in the cross-section, select the arc to modify, and then make changes on the ribbon.

Modify an Arc in a Path (on page 102)

6. To modify a turn in the cross-section, select the turn to modify, and then make changes on the ribbon.

Modify a Turn in a Path (on page 102)

7. To move a segment in the cross-section, click the segment to move, and then click the point from which to move the segment and the point to which to move the segment.

Move Segments of a Path (on page 99)

8. To add segments to the cross-section, click **Create** on the ribbon, and then click to place the new segments.

Add Segments to a Path (on page 98)

TIP You can add as many segments to the cross-section as you need. However, you must close the cross-section to be able to save it.

9. To delete a segment in the cross-section, select the segment to delete, and then click **Delete Selected Items** .

10. Click **Finish Path**.

TIPS

- The **Finish Path** button is not available until you close the cross-section by clicking the starting point.
- The new shape appears in dynamics when you click **Finish Path**.
- The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.

Edit cross-section properties for a prismatic shape

1. Click **Select** .
2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.
TIP If the **Locate Filter** box is not displayed on the horizontal ribbon, click the **Select Tool**  on the vertical toolbar.
3. On the ribbon, click **Properties** .
4. In the **Prismatic Shape Properties** dialog box, click the **Cross-Section** tab.
5. Make modifications to the cross-section properties.
TIP As you make changes to the cross-section properties, the cross-section changes appear dynamically in the model.
6. Click **OK**.

- Click **Finish** to apply the changes to the cross-section to the volume.

NOTES

- If you sketch a cross-section, the properties for the cross-section include the X-, Y-, and Z-coordinates, the turn type, and values for each point that define the cross-section path.
- You cannot modify the properties of a sketched cross-section in the **Prismatic Shape Properties** dialog box. Instead, you must make changes to the path that defines the cross-section sketch. For more information, see *Modify a sketched cross-section* (on page 96).

Add segments to a path

- Click **Select** .
- On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.
- On the ribbon, click **Path** .
- TIP** The software highlights the path.
- To add one or more segments to the path, click **Create**.
- Click the point on the existing path to insert the new segment.



TIPS

- You can use **PinPoint**, **Point Along**, and the SmartSketch relationship indicators when defining your path.
- You can change the plane for the path at any time by selecting the plane in the **Plane** list.
- To change the segment from a straight line to an arc, click **Arc by 3 Points**  in the **Path Type** list, and then click three points to define the arc.
- To switch back to a straight line after sketching an arc, click **Line**  in the **Path Type** list.
- To break the path, click **No Line**  in the **Path Type** list.

- Click to place other points and add to the path as needed.



TIP To change the turn type for the corner, click the new turn type in the **Turn Type** list and define the angle or dimensions for the turn in the **Feature Value** box.

- After you place all the points for the new segments, click **Finish Path**.



NOTES

- The software does not require that you close the path. You can end the path at any point.
- You can click **Edit** on the ribbon to change the segment or modify the path further.
- To manually set the length and angle for a segment of the path, change the values in the **Angle** and **Length** boxes on the ribbon.

See Also

Creating Customized Shapes (on page 80)

Move Segments of a Path

- Click **Select** .
- On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.



3. On the ribbon, click **Path** .
- The software highlights the path.*
4. Select the segments to move.



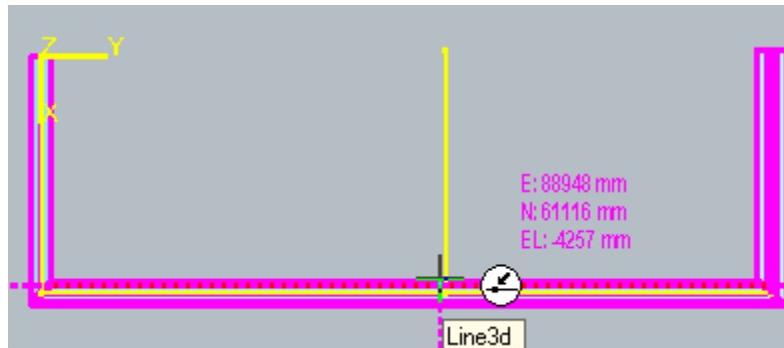
TIP You can select multiple segments by holding the CTRL key and clicking the segments.

5. To keep the length of a straight segment constant while you move the segment, click **Length Locked** .

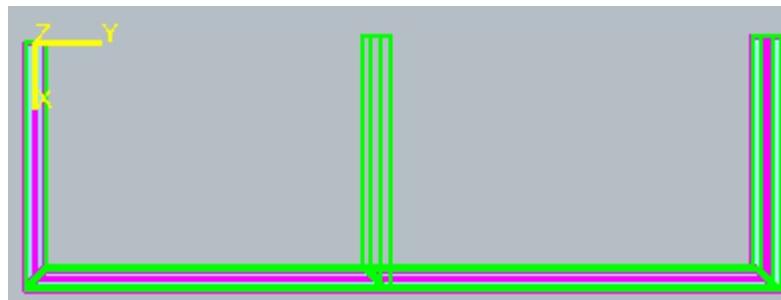
TIPS

- When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of the adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.
- When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.

6. Click **Move From**  to specify the starting location of the move vector.
7. Click to specify the ending location of the move vector.

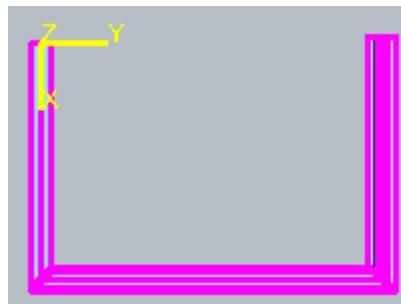


8. Click **Finish Path**.



!TIPS

- The new shape appears in dynamics when you click **Finish Path**. The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.



- While modifying several elements one at a time, you must use CTRL to select the next element, and then CTRL to de-select the previous element.

Modify a straight segment in a path

1. Click **Select**
 2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.
 3. On the ribbon, click **Path** .
- The software highlights the path.*
4. Select the straight segment to modify.
- !TIP** You can select multiple segments by holding CTRL and clicking the segments.
5. To change the line type for the segment, click a new type in the **Path Type** list.
 6. To change the plane for the segment, click a new plane in the **Plane** list.
 7. To delete the segment, click **Delete Selected Items** .
 8. To keep the length of a straight segment constant when you move the segment, click **Length Locked** .

◆ TIPS

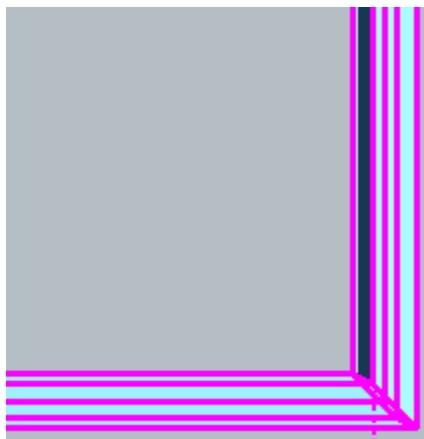
- When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of the adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.
 - When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.
9. Click **Finish Path**.
- ◆ TIP** The new shape appears in dynamics when you click **Finish Path**. The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.
- ◆ NOTE** You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 99).

Modify an Arc in a Path

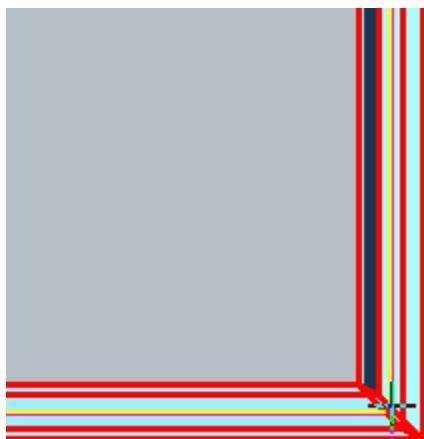
1. Click **Select** .
 2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.
 3. On the ribbon, click **Path** .
 - The software highlights the path.*
 4. Select the arc to modify.
- ◆ TIP** You can select multiple segments by holding CTRL and clicking the segments.
5. To change the line type for the segment, click a new type in the **Path Type** list.
 6. To change the plane for the segment, click a new plane in the **Plane** list.
 7. To delete the segment, click **Delete Selected Items** .
 8. Click **Finish Path**.
- ◆ TIP** The new shape appears in dynamics when you click **Finish Path**. The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.
- ◆ NOTE** You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 99).

Modify a turn in a path

1. Click **Select** .
2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.



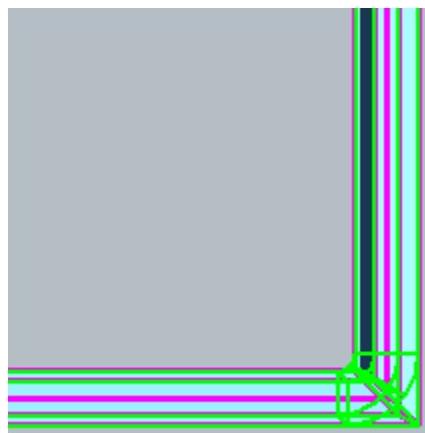
3. On the ribbon, click **Path** .
- The software outlines the path in yellow.*
4. Select the turn to modify.



 **TIP** When the pointer passes over an object that contains multiple elements, use the **QuickPick** feature to help you select the specific element.

5. To change the plane for the segment, click a new plane in the **Plane** list.
6. To delete the turn, click **Delete Selected Items** .
7. To change the turn type, select a new type in the **Turn Type** list.
8. To change the dimensions for the selected turn type, type a value in the **Feature Value** box.

9. Click **Finish Path**.



TIP The new shape appears in dynamics when you click **Finish Path**. The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.



NOTE You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 99).

Sketch Properties Dialog Box

Sets options for paths. You cannot edit the different applicable properties.

See Also

General Tab (Sketch Properties Dialog Box) (on page 105)

General Tab (Sketch Properties Dialog Box)

Displays the points that make up a path, their X-, Y-, and Z-coordinates, turn types, and turn type dimensions.

Point No

Displays the point number that identifies the selected point.

X

Displays the location of the point on the X-axis.

Y

Displays the location of the point on the Y-axis.

Z

Displays the location of the point on the Z-axis.

Turn Type

Displays the type of turn associated with the point. Turn types include none, bend, chamfer, and cornice.

Value

Specifies dimensions for the selected turn type. For bends and cornices, the value specifies the radius of the bend. For chamfers, the value specifies the dimensions for setback A and setback B of the chamfer.

SECTION 9

Place Nozzle

 Adds different types of ports to an equipment or equipment component object. You can specify port location details and properties, including port type (such as foundation ports). The ability to add ports to an equipment object is required to connect equipment to distributed systems, such as conduit, HVAC, piping, and cable.

Placing Foundation Ports

When you use the **Place Nozzle** command to place a foundation port, you are defining the bolt hole pattern, hole information, and mounting plane for the equipment. A piece of equipment can have more than one foundation port. During placement, the X-, Y-, and Z-axis of the port displays so you can place the foundation port precisely in the model.

Place Nozzle Ribbon

Sets options for defining port (such as foundation ports) properties and selecting the shape object with which the port is associated.

Nozzle Properties

Displays the dynamic occurrence properties so that you can review the static properties of any port attached to an existing shape.

Port/Nozzle Parent

Specifies the parent (which can be a shape, equipment, or an equipment component by way of their coordinate system) with which the port is associated.

Nozzle Name

Specifies the nozzle name.

Nozzle Length

Sets the nozzle length.

Nozzle Bend Length

Sets the nozzle bend length.

Nozzle Bend Radius

Sets the nozzle bend radius. The **Bend Radius** value should be less than or equal to the **Nozzle Bend Length** and **Nozzle Length**.

NOTES

- The **Nozzle Length** property is available only for elbow nozzles, piping straight nozzles, and HVAC nozzles.
- The **Nozzle Bend Length** property is available only for elbow nozzles.
- The **Nozzle Bend Radius** property is available only for elbow nozzles.

What do you want to do?

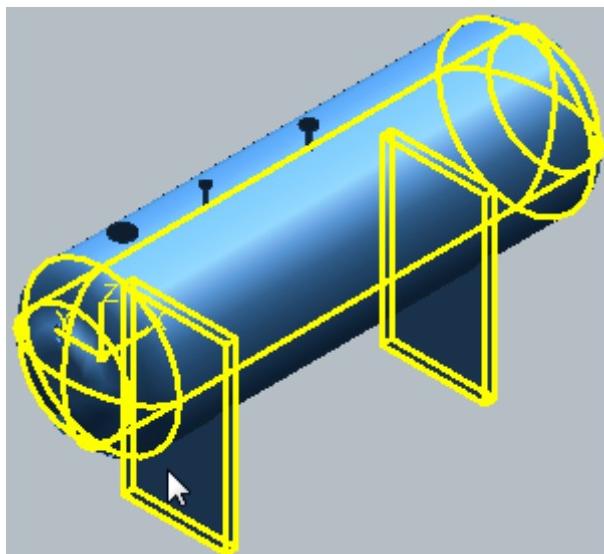
- *Place a nozzle* (on page 107)
 - *Place a nozzle from a P&ID* (on page 109)
 - *Place a foundation port* (on page 110)
-

Place a nozzle

1. Click **Place Nozzle**  on the vertical toolbar.

!TIP The nozzle is connected to the cursor to identify the placement location.

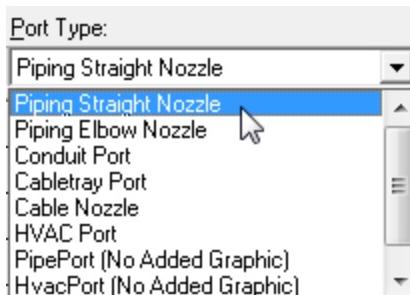
2. Select the parent to which to add a nozzle.



!TIP You can either select the parent in a graphic view, or you can select it in the **Workplace Explorer**.

The **Nozzle Properties** dialog box appears.

3. On the **Occurrence** tab, select the type of nozzle to place from the **Port Type** list.



TIP If you select the **PipePort (No Added Graphic)** or **HvacPort (No Added Graphic)** type, see the **Note** section following this procedure for information regarding its placement.

- In the **Value** grid, type values for the listed properties.

TIPS

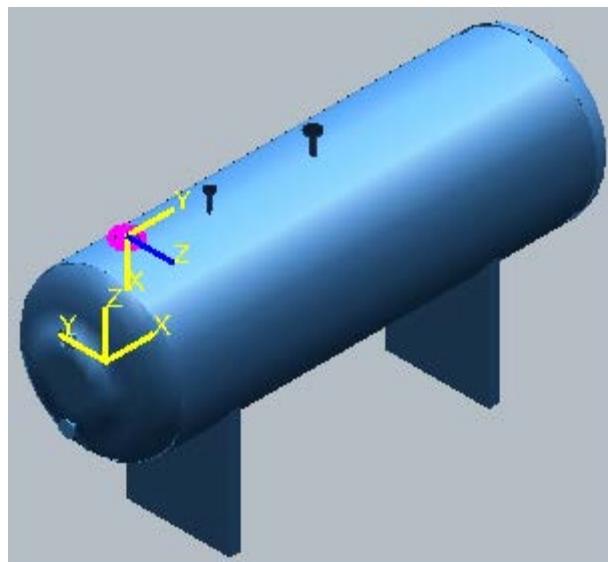
- You can use the **From catalog** button to pre-populate values from predefined catalog nozzles. Values can be overridden after selection of predefined values.
- You can use the **Category** list to control the types of properties that display in the dialog box.
- By default, insulation for the nozzle is set the same as the equipment object on which it is placed. If necessary, you can edit insulation properties during initial placement of the nozzle, as well as after it is placed. For more information, see the **Insulation and Tracing** section on the *Occurrence Tab (Nozzle Properties Dialog Box)* (on page 111).

- Click the **Location** tab, and select a placement type from the **Placement Type** list box.

- In the **Value** grid, enter the values for the listed properties.

TIP These location properties are dimensions relative to the selected parent.

- Click **OK**.



The **Nozzle Properties** dialog box closes, and you return to the model with the nozzle placed, unless the **Placement Type** selected is **Position by Plane and Axis** or **Position by Point**. In this case, additional steps are required to place the nozzle in the model.

NOTES

- The **PipePort (No Added Graphic)** or **HvacPort (No Added Graphic)** port type is a special implementation of a piping straight nozzle that is used to add a nozzle port to an existing graphic. Equipment objects imported with the **Place Imported Shape from File** command may have a graphic representation of a nozzle, but no nozzle port. The **PipePort (No Added Graphic)** or **HvacPort (No Added Graphic)** nozzle type, which has all the standard piping port properties but not the graphics of the piping nozzle, allows you to add a nozzle port (represented as a circular surface) to the existing shape. After selecting **PipePort (No Added Graphic)** or **HvacPort (No Added Graphic)**, the software prompts you to graphically

locate the port in the model by selecting a surface and a cylinder (cone) to define a position and an axis. You can also place **PipePort (No Added Graphic)** or **HvacPort (No Added Graphic)** using other placement types.

- Press the left or right arrow keys to rotate the nozzle by 90-degree increments at any time during the placement of the nozzle. Press the up arrow to scroll through the three possible axes of rotation. Press the down arrow key to scroll to each datum point.

Place a nozzle from a P&ID

1. Click **SmartPlant > Retrieve** to retrieve the P&ID that contains the nozzle to place. You can skip this step if the P&ID has already been retrieved.
2. Click **SmartPlant > View P&ID** to view the P&ID that contains the nozzle to place.
3. In the P&ID, select the nozzle to place in the model.
4. Click **Place Nozzle**  on the vertical toolbar.
5. Select the equipment or designed equipment object on which to place the nozzle.

TIP You can either select the shape in a graphic view, or you can select it in the **Workplace Explorer**.

The **Nozzle Properties** dialog box appears.

6. Verify that the property values defined on the **Occurrence** tab are correct as compared to the nozzle report.
7. On the **Location** tab, specify the placement type and type values for the location parameters as needed.
8. Click **OK**.

The **Nozzle Properties** dialog box closes, and you return to the model with the nozzle placed.

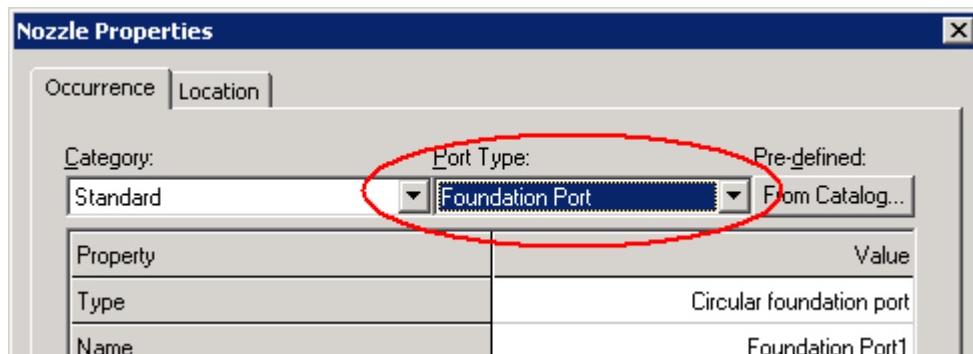
9. Click **SmartPlant > Compare Design Basis**, and verify the correlation status of the nozzle in the model.
10. If necessary, click **Copy properties from design basis**, and then click **Update** to transfer data in the P&ID to match the 3D model object.

NOTES

- The software automatically sets all mapped nozzle properties in the 3D model as defined by the currently selected P&ID nozzle.
- After the nozzle is placed in the 3D model, it can be moved without affecting the correlation. If it is deleted, it does not delete the P&ID design basis object.
- Correlation is a relationship between an object in the 3D model and the corresponding object in the P&ID. For example, a nozzle in the 3D model is related to a nozzle in the P&ID.
- The occurrence properties of the nozzle object in the 3-D model that are not correlated with corresponding properties in the design basis object (that is, the nozzle in the P&ID) can be edited without changing the correlation status.

Place a foundation port

1. Click **Place Nozzle**  on the vertical toolbar.
2. Select the parent to which to add a nozzle.
*The **Nozzle Properties** dialog box appears.*
- TIP** You can either select the parent in a graphic view, or you can select it in the **Workplace Explorer**.
3. On the **Occurrence** tab, select **Foundation Port** as the type of nozzle to place from the **Port Type** list.



The properties and values update to reflect those for foundation ports. For more information, see Foundation Port Properties Dialog Box (on page 121).

4. On the **Occurrence Tab**, in the **Value** grid, type values for the listed properties.
TIP The **Type** property selection updates the other properties in the grid. For more information on the property changes per type, see *Occurrence Tab (Foundation Port Properties)* (on page 121).
5. Click the **Location** tab, and select a placement type from the **Placement Type** list box.
NOTE If you select the **Position by Point** placement type, the z-axis appears with the foundation port during placement for more precise orientation with the equipment object.
6. In the **Value** grid, type the values for the listed properties.
TIP These location properties are dimensions relative to the selected parent.
7. Click **OK**.

*The **Nozzle Properties** dialog box closes, and you return to the model to place the foundation port*

NOTES

- Press the left or right arrow keys to rotate the foundation port by 90-degree increments at any time during the placement operation. Press the up arrow to scroll through the three possible axes of rotation. Press the down arrow key to scroll to each datum point.
- To edit the properties of the new foundation port, right-click the foundation port and select **Properties**.

Nozzle Properties Dialog Box

Displays nozzle properties for review and editing.

See Also

- Occurrence Tab (Foundation Port Properties) (on page 121)*
- Occurrence Tab (Nozzle Properties Dialog Box) (on page 111)*
- Location Tab (Nozzle Properties Dialog Box) (on page 120)*
- Configuration Tab (on page 148)*

Occurrence Tab (Nozzle Properties Dialog Box)

Displays the nozzle properties that you can edit. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. You can modify these values after the initial placement of the nozzle to reflect exact design needs or deviations from the standard part. If you selected more than one nozzle and then selected the properties command, only the common properties between the selected nozzles appear.

Category

Displays the defined category name for the selected nozzle. This data is retrieved from the Equipment and Furnishings reference data.

Port Type

Specifies the type of port for placement on a selected shape. If the selected port type requires you to define physical dimensions, a picture of the selected port type displays. The picture contains labeled parameters, such as A, B, C, and so forth, that you can define in the **Value** column of the grid. If the selected port type requires no user-defined physical dimensions, no picture displays. This option does not display when you edit a nozzle that you have already placed in the model.

From catalog

Displays the **Select Nozzle** dialog box so that you can select an existing nozzle from the Catalog.

The following is a complete list of properties, both common and unique, that can be defined using the **Occurrence** tab. Initial values are defined in the reference data. For more information on properties, refer to the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

NOTES

- Properties that display on the **Occurrence** tab are dependent on the port type you select.
- Properties are listed alphabetically, rather than in the order in which they may appear.

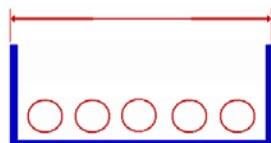
Actual Depth

Specifies the actual depth of the cable tray port.



Actual Width

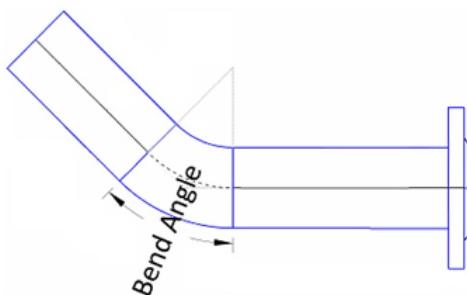
Specifies the actual width of the cable tray port.

**Area**

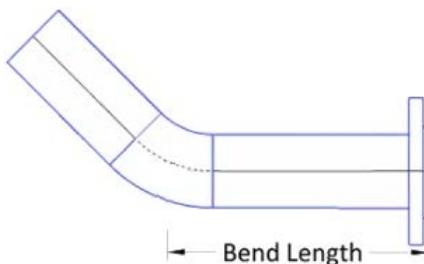
Specifies the diameter of the object opening.

Bend Angle

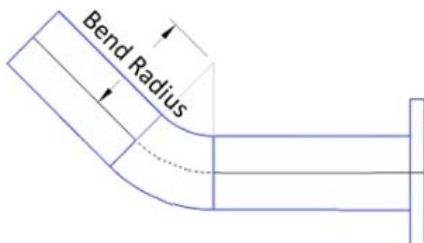
Specifies the required angle for an object.

**Bend Length**

Specifies the allowable bend length as an absolute value.

**Bend Radius**

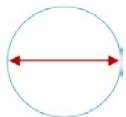
Specifies the allowable measurement for the bend radius.

**Behavior Controlled by User**

Indicates whether or not you can delete the object. Select **False** to prevent anyone from deleting the object from the model.

Conduit Diameter

Specifies the diameter of the conduit.

**Corner Radius**

Specifies the allowable measurement for the corner radius.

Cpt Offset

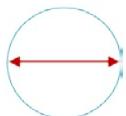
Displays the offset value of the center point for the object.

Cross Section Shape

Displays the shape of the object cross section.

Diameter

Displays the diameter of the object.

**Dimension Base Outer**

Displays the outer dimension of the base of the nozzle.

End Practice

Specifies the end practice for the object. To change the options on the list, edit the **End Standard** select list in Catalog.

End Preparation

Specifies the end preparation code for the part port. To change the options on the list, edit the **End Preparation** select list in Catalog.

End Standard

Specifies the end standard code for the part port. To change the options on the list, edit the **End Standard** select list in Catalog.

Flange or Hub Outside Diameter

Specifies the diameter for the flange or hub of the object.

**Flange or Hub Thickness**

Specifies the thickness of the flange or hub of the object.



Flange or Mechanical Groove Width

Specifies the width of the flange or mechanical groove of the object.

Flange Projection or Socket Offset

Specifies the projection of the flange or offset of the socket for the object.

**Flange Width**

Specifies the width of the flange of the object.

Flow Direction

Specifies the flow direction code for the port. Specifying the correct flow direction code for each part port is very important because the software automatically orients the part to the flow direction of the pipe or port to which the part is connected. To change the options on the list, edit the **Flow Direction** select list in Catalog.

Hub Outside Diameter

Displays the outside diameter of the hub for the object.

Liner Thickness at Face of Flange

Specifies the thickness of the flange face for the object.

Liner Thickness at Inside Diameter

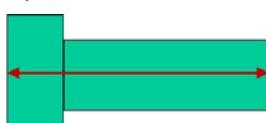
Specifies the thickness of the inside diameter of the object.

Name

Specifies the name of the object. If a **Name Rule** is specified, then the software uses that rule to determine this name. If the **Name Rule** value is **User Defined**, then you must type a name in this box.

Nozzle Length

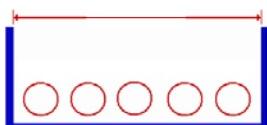
Specifies the measurement for the nozzle length.

**Nominal Size**

Specifies the nominal diameter (NCD) of the object. If displays in the box, the NCD for the object is inherited from the parent object to which the object belongs. If the icon does not display, and you need to use the parent object specification from the parent object run, select **Use Run NCD**. If you have manually selected the part, the NCD of the selected part displays, and you cannot change it.

Nominal Width

Specifies the nominal width of the cable tray port.

**Nominal Depth**

Specifies the nominal depth of the cable tray port.

**NPD Unit Type**

Specifies the units for the **NPD** value. For example, type **mm** or **in**.

Piping Outside Diameter

Specifies the outside diameter of the piping for the nozzle.

**Piping Point Basis**

Specifies the piping point basis code that identifies the function of the port. Valid codes are listed in the **AllCodeLists.xls** workbook on the **Piping Point Basis** sheet in the **Codelist Number** column.

Port Depth

Specifies the depth of the port.

Port Index

Specifies the index number for the port.

Port Type

Specifies the electrical port type of the nozzle.

Pressure Rating

Specifies the pressure rating code for the part port. Valid codes are listed in the **AllCodeLists.xls** workbook on the **Pressure Rating** sheet in the **Codelist Number** column.

Raised Face or Socket Diameter

Specifies the diameter of the raised face or socket of the nozzle.

Rating Practice

Specifies the rating practice for the nozzle.

Reinforced Wall Thickness

Specifies the thickness of the reinforced wall of the nozzle.

Schedule Practice

Specifies the schedule practice for the nozzle.

Schedule Thickness

Specifies the schedule thickness short description or code for the part port. To change the options on the list, edit the **Schedule Thickness** select list in Catalog.

Seating or Groove or Socket Depth

Specifies the depth of the seating, groove, or socket of the nozzle.

**Shape Depth**

Displays the depth of the nozzle shape.

Shape Width

Displays the width of the nozzle shape.

Sub Type

Specifies the electrical sub-type of the nozzle.

SuppressGraphics

Specifies whether or not to display the nozzle graphics. Select **True** to hide the nozzle graphics. By default, this property is set to **False**. **SuppressGraphics** only displays on pipe nozzles.

Terminal

Specifies the type of terminal electrical connection for the port.

Termination Class

Specifies the termination class for the nozzle. The list is defined by the **TerminationClass** codelist.

Termination Subclass

Specifies the termination sub-class for the nozzle. The list is defined by the **TerminationSubClass** codelist.

Thickness

Specifies the thickness of the object.

Tightness

Specifies the tightness of the nozzle.

Wall Thickness or Groove Setback

Specifies the thickness of the wall or the setback of the groove for the nozzle.

Position and Orientation

NOTE The **Position and Orientation** properties are editable only if the **Placement Type** is set to **Position by Point**.

East

Displays the distance of the connection point from the active coordinate system origin in the east direction.

North

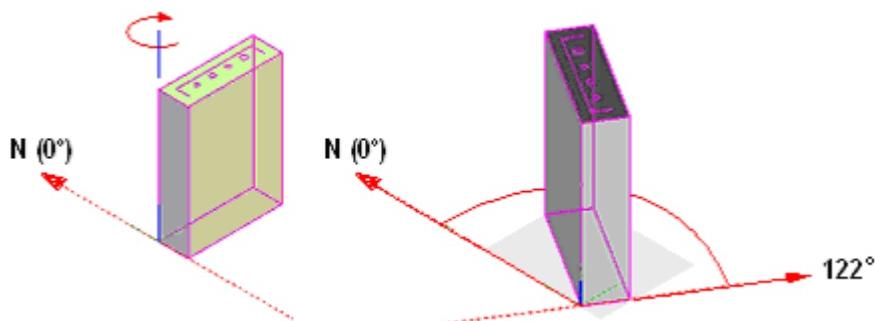
Displays the distance of the connection point from the active coordinate system origin in the north direction.

Elevation

Displays the distance of the connection point above or below the active coordinate system origin.

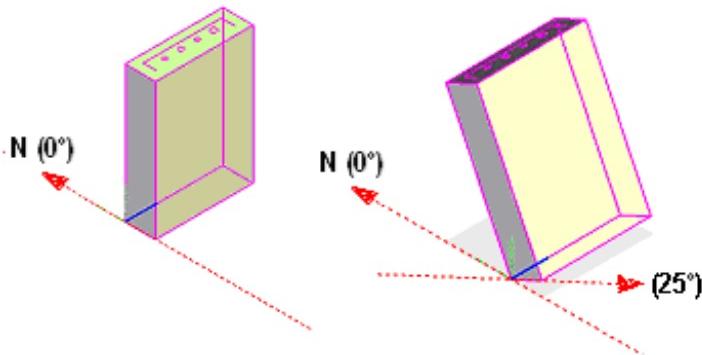
Bearing

Specifies the bearing angle for the object. The **Bearing** angle is measured between the local x-axis of the object and the Y-axis (North) of the global coordinate system in the XY-plane. The local x-axis is the default axis of primary symmetry for all symbols in the catalog. The bearing measurement direction is clockwise from the active coordinate system North looking in the negative active coordinate system direction; that is, down from 0 to 360 degrees. You can enter negative bearing angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the bearing measure displays 0 degrees.



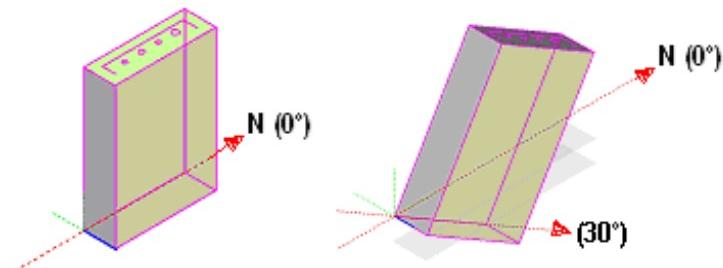
Pitch

Specifies the pitch angle for the shape. The **Pitch** angle measures between the x-axis of the object and the X-axis of the global coordinate system in the XZ-plane. This option sets the reference in the model to a line that is the intersection of the vertical plane through the X-axis of the local coordinate system and the active coordinate system horizontal plane. The angle is measured in the positive direction from the horizontal plane in the active coordinate system up direction regardless of the current bearing. Another way to describe pitch is the rotation of the object about its y-axis. Pitch angles are limited to between -90 degrees and +90 degrees, with 0 indicating horizontal.



Roll

Specifies the roll angle for the object. The **Roll** angle measures between the local z-axis of the object and the Z-axis of the global coordinate in the YZ-plane. Another way to describe roll is the rotation of the object about its x-axis. This option sets the reference in the model to a line that is perpendicular to the local coordinate system x-axis and in the horizontal plane. Roll angles are measured clockwise from horizontal to the y-axis of the local coordinate system. The roll angle is between 0 and 360 degrees. You can enter negative roll angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the reference in the model is the North axis.



Insulation and Tracing

Insulation Criteria

Specifies the insulation criteria for the nozzle.

Up to Neck Only indicates that the nozzle is insulated up to the neck, leaving the flange uninsulated

Complete indicates that both the nozzle neck and flange are insulated.

The list is defined by the InsulationCriteria codelist.

Insulation Purpose

Specifies the purpose of insulation.

Insulation Material

Specifies the material of insulation.

Insulation Thickness

Specifies the thickness of insulation material.

Is Insulated

Specifies whether the nozzle object is insulated. If set to **False**, you cannot modify the information to any of the insulation properties.

Insulation Operating Temperature

Specifies the operating temperature.

Insulation Surface Area

Specifies the measurement of the surface area of insulation.

Insulation Requirement

Specifies the requirement of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Insulation Type

Specifies the type of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

NOTES

- If  displays in the **Value** box, the corresponding insulation property for the object is inherited from the parent equipment or equipment component to which the object belongs.

Insulation Thickness	0.04 m	
Is Insulated	True	

- Insulation is supported only on piping and HVAC equipment and component nozzles.

Select Nozzle Dialog Box

Specifies the nozzle needed for placement. This dialog box appears when you click **From Catalog** on the **Nozzle Properties** dialog box. By browsing through the part hierarchy, you can find any nozzle in the Catalog database. After you select a nozzle, the software returns you to the **Nozzle Properties** dialog box.

 **Back**

Returns you to the previously selected nozzle. Use this command to navigate through the hierarchy to the specific part that you need.

 **Forward**

Sends you to the last selected nozzle that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific part that you need.

 **Up One Level**

Brings up the next highest level of the Catalog hierarchy. Use this command to navigate through the hierarchy to the specific part that you need.

 **Properties**

Displays the nozzle properties as defined in the catalog.

 **Preview**

Displays a bitmap symbol of the selected nozzle. The image file must be assigned to the nozzle in the catalog reference data.

 **List View**

Sets the dialog box to display nozzles in a list view.

 **Grid View**

Sets the dialog box to display nozzles in a spreadsheet-style grid view.

Address

Specifies your exact location within the displayed hierarchy.

See Also

Occurrence Tab (Foundation Port Properties) (on page 121)

Occurrence Tab (Nozzle Properties Dialog Box) (on page 111)

Location Tab (Nozzle Properties Dialog Box)

Sets options for placing a nozzle on a designed equipment object.

Placement Type

Specifies a location type for the nozzle. The following location types are available for nozzles:

Placement Type	Description
Radial	Straight nozzle with nozzle centerline on the radial plane.
Tangential	Straight nozzle on the radial plane, but the nozzle centerline does not intersect the vessel axis.
Axial	Straight nozzle normal to radial plane.
Skew	Straight nozzle that is tilted from both the radial plane and the vessel axis.
Axial Elbow	Elbow nozzle with hub centerline normal to the radial plane.
Radial Elbow	Elbow nozzle placed with the hub radial to the vessel axis.
Offset Radial Elbow	Elbow nozzle placed with the hub tangential to the vessel axis.

Placement Type	Description
Offset Skew	Skew nozzle with a parallel offset in the radial plane.
Position by Plane and Axis	Available with all Nozzles including PipePort (No Added Graphic) and HvacPort (No Added Graphic).
Position by Point	Available with all Nozzles including PipePort (No Added Graphic) and HvacPort (No Added Graphic).

Property

Identifies the location parameters of the selected nozzle.

Value

Shows the current values for all location parameters of the selected nozzle. You can modify these values after the initial placement of the nozzle to reflect exact design needs or deviations from the standard part.

Foundation Port Properties Dialog Box

Displays foundation port properties for review and editing.

See Also

- Location Tab (Foundation Port Properties)* (on page 123)
- Holes Tab (Foundation Port Properties)* (on page 124)
- Occurrence Tab (Foundation Port Properties)* (on page 121)
- Configuration Tab* (on page 148)

Occurrence Tab (Foundation Port Properties)

Displays the foundation port properties that you can edit.

Category

Displays the defined category name for the selected nozzle. This data is retrieved from the Equipment and Furnishings reference data.

Port Type

Indicates that the port type is **foundation port** and is read-only.

From catalog

This button is not available when editing properties for a foundation port.

General Properties for All Foundation Ports

Type

Specifies the type of foundation port. The available types are **Rectangular Foundation Port**, **Circular Foundation Port**, **Single Hole**, and **No Hole**. The **Type** changes the other properties available on the foundation port.

Name

Specifies the name applied to the selected foundation port. Foundation ports do not require naming rules. If a naming rule is not applied, a default naming convention is applied, such as the equipment name with the foundation port number appended (for example: TankA-FoundationPort1).

Default Foundation

Specifies a default structural foundation for the equipment. For more information on default foundations for specific equipment objects, see the Equipment.xls file.

Liner Thickness

Specifies the thickness of the liner for the foundation port.

Footprint Offset

Specifies the offset value to be used by Structure when the software constructs the foundation for the foundation port. Smart 3D does not currently use this property.

Hole Diameter

Specifies the diameter of the bolt holes.

Behavior Controlled by User

Indicates whether or not you can delete the object. Select **False** to prevent anyone from deleting the object from the model.

Circular Foundation Port Type

When you select the **Circular Foundation Port** type, the software generates an equally spaced circular bolt hole pattern based on the **Number of Holes** and the **Bolt Circle Diameter** properties.

Additional properties are:

Number of Holes

Specifies the number of bolt holes for the foundation port.

Bolt Circle Diameter

Specifies the diameter of the bolt holes.

Rectangular Foundation Port Type

When you select the **Rectangular Foundation Port** type, the software uses "bolt hole 1" as the foundation port origin with the y-axis pointing to "bolt hole 2".

Additional properties are:

Number of Bolt Holes in X

Specifies the number of bolt holes on the x-axis of the foundation port.

Number of Bolt Holes in Y

Specifies the number of bolt holes in the y-axis of the foundation port.

Distance between Holes (X)

Specifies the distance between the bolt holes on the x-axis of the foundation port.

Distance between Holes (Y)

Specifies the distance between the bolt holes on the y-axis of the foundation port.

Single Hole Type

When you select the **Single Hole** foundation port type, the software places the local coordinate system origin at the center of the bolt hole. The software does not generate a default footprint boundary with this option.

No Hole Type

When you select the **No Hole** foundation port type, the software generates a default footprint boundary and places the local coordinate system origin for the foundation port at the lower left with the y-axis pointing up.

NOTE If the properties on this dialog box are read-only, the selected foundation port is part of the delivered equipment object.

Location Tab (Foundation Port Properties)

Sets options for a foundation port on an equipment object.

Placement Type

Specifies a location type for the nozzle. The following location types are available for foundation ports:

Placement Type	Description
Radial	Straight foundation port with centerline on the radial plane.
Tangential	Straight foundation port on the radial plane, but the port centerline does not intersect the vessel axis.
Axial	Straight foundation port normal to radial plane.
Skew	Straight foundation port that is tilted from both the radial plane and the vessel axis.
Offset Skew	Skew foundation port with a parallel offset in the radial plane.
Position by Point	Available with all foundation ports.
Position by Plane and Axis	Available with all foundation ports.

Property

Identifies the location parameters of the selected foundation port.

Value

Shows the current values for all location parameters of the selected foundation port. You can modify these values during the initial placement of the foundation port to reflect exact design needs or deviations from the standard part.

Holes Tab (Foundation Port Properties)

Displays X- and Y-location, diameter, and mounting information for each bolt hole.

This information is read-only.

SECTION 10

Place Imported Shape from File

 Adds geometry to a designed equipment object that was modeled with solid modeling software and saved to a VUE, SAT, or MicroStation DGN file format to an equipment or equipment component object. The **Place Imported Shape from File** command uses the equipment or equipment component object selected in the **Workspace Explorer** hierarchy. If no equipment or equipment component object is selected prior to selecting the command, you are prompted to select an equipment or equipment component object.

The command creates an *Imported Shape* object with the defined properties and a local coordinate system. The imported shape coordinate system is defined by the VUE, SAT, or MicroStation file. The new imported shape does not retain a connection to the original import file.

After importing a shape, you can use the **Place Nozzle** command to complete the equipment definition. For more information, see *Place Nozzle* (on page 106) in the *Equipment and Furnishings User's Guide*.

You can also edit the shape properties. For more information, see *Edit Shape Properties* (on page 75).

 **NOTE** The **Place Imported Shape from File** command supports SAT files of ACIS R22 and earlier.

Select Shape File Dialog Box

Specifies the VUE, SAT, or MicroStation DGN file that contains the geometry to import as the shape. This dialog box appears automatically when you click the **Place Imported Shape from File**  command. You can navigate through the available file systems to locate the appropriate file. After you select a file and click **OK**, the **Display Aspects** dialog box appears so you can select the aspect(s) for the imported shape. The software returns you to the model to place the imported shape.

When you use **Place Imported Shape from File**  to place an imported shape from a .sat, .dgn, or .vue file on the symbol share, Smart 3D remembers the relative path to the file. This lets you share design files between models during **Copy/Paste** operations, **Model Data Reuse**, **Paste from Catalog**, and **Mirror Copy** of imported shapes. If you import a shape file from a location other than the symbol share, Smart 3D remembers the full path to that file. In both cases, the full path displays on the **Properties** dialog box for the shapes. For more information, see *Share imported shapes between models* (on page 128).

 **NOTE** To save the relative path to imported shapes placed with earlier versions of Smart 3D, you must remodel those shapes.

You can also right-click the new shape object and select **Properties** to further define the object. Use **Place Nozzle**  to add nozzles and/or ports to the object.

 **NOTE** The **Select Shape File** dialog box is a standard Microsoft Windows dialog box. For information regarding how to use the dialog box, click the question mark  in the upper right corner, and then click a control on the dialog box.

Place an Imported Shape

1. Click **Place Imported Shape from File**  on the vertical toolbar.

! TIP If an equipment or equipment component object has not been selected, select one in a graphic view or in the **Workspace Explorer**.

*The **Select Shape File** dialog box appears.*
2. Navigate to and select the VUE, SAT, or MicroStation DGN file that contains the shape to insert, and then click **OK**.
3. In the **Display Aspect** list box, select the aspects from the list.

! TIP You can click **Format > View** and change the **Render Selected Aspects** option in the **Format View** dialog box to display the aspect in the active graphic view.
4. Click **OK**.

The list box closes, and you return to the model.
5. Click in the graphic view to select an approximate location or reference element for a relationship for the shape.

NOTE After a shape has been placed in the model as part of the equipment or equipment component object, you can use the horizontal ribbon to add or change the positioning relationship. You can add port definitions with the **Place Nozzle**  command to complete the equipment or equipment component definition.

SAT File Limitations

- From a performance point of view, the SAT file size should be less than 5 MB.
- From a drawings point of view, the files must not be detailed and must not spread across a large space as they are caught in volume filters for Drawings.
- Spreading the SAT files across multiple equipment reduces the volume of each .sat file.
- For efficient Interference Checker processing, SAT files should not have a footprint (range) greater than 100 meters.
- When we import the SAT files created using a third-party tools, they are converted to Intergraph GType format. In some cases, the generated objects may not be valid after the dual conversion, such as when you use AutoCAD to convert an object to ACIS SAT format, and then convert it to GType. The possible causes for this failure are:
 - You are using an old version of the SAT file.
 - The units of measurement do not match. For example, scaling can result in a degenerate geometry if the distance between two points becomes less than 1.0E-6 M.
 - The orientation of the boundary curves is inconsistent with the surface normal. For example, when applying the right-hand rule, the object is on your left when moving in the direction on the boundary, and the surface normal is upward.
- The valid geometries that you can use in creating SAT files are listed below.
 - GLineTYPE
 - GEllipseTYPE

- GArcTYPE
- GLineStringTYPE
- GBspCurveTYPE
- GComplexStringTYPE
- GBspSurfaceTYPE
- GTorusTYPE
- GSphereTYPE
- GConeTYPE
- GTippyConeTYPE
- GPlaneTYPE
- GShapeTYPE
- GProjectionTYPE
- GPipeTYPE
- GRevolutionTYPE
- GEbowTYPE
- GRuledTYPE
- GRuledNormalTYPE
- GValveTYPE
- GReducerTYPE
- GPolyMeshTYPE
- GPointTYPE
- GVolumeTYPE
- GText3dTYPE

VUE File Limitations

- You cannot import the shape from the VUE file under a design solid.
- The VUE file size should be less than 10 MB. If the VUE file size exceeds 10 MB, the software displays a warning message.
- If the VUE file size exceeds 25 MB, the software displays an error message and you cannot place the object.
- The imported graphics are not smart graphics. They do not contain properties.
- The valid geometries that you can use are the same as those you can use in creating SAT files.

Share imported shapes between models

For imported shapes placed from a file on the symbol share, Smart 3D remembers the relative path to that file. If you import a shape file from a location other than the symbol share, Smart 3D remembers the full path to that file. In both cases, the full path displays on the **Properties** dialog box for the shapes.

When you place these shapes in the model using a file on the symbol share, and then later use **Copy/Paste** operations, **Model Data Reuse**, **Paste from Catalog**, or **Mirror Copy**, then the shapes will refer to a file on the target model symbol share rather than the source model symbol share. If the shapes in the source model are placed using a file from somewhere other than the symbol share, then during the **Copy/Paste** operations, **Model Data Reuse**, **Paste from Catalog**, or **Mirror Copy**, the shapes will refer to that physical path.

NOTES

- During **Model Data Reuse**, **Copy/Paste**, or **Paste from Catalog**, the software verifies that the target file exists. If the target file does not exist in the path, Smart 3D does the following:
 - Copies the graphics of the source model imported shape.
 - Pastes the same graphics in the target model imported shape.
 - Adds an item to the **To Do List** indicating that the symbol failed because the imported shape source file is not found in the path. This applies to shapes imported from shared content as well as shapes imported from a physical location.
- If the software detects a problem during **Model Data Reuse**, **Copy/Paste**, or **Paste from Catalog**, such as an empty file or a file that it cannot convert, then Smart 3D does the following:
 - Copies the graphics of the source model imported shape.
 - Pastes the same graphics in the target model imported shape.
 - Adds an item to the **To Do List** with an appropriate message.

SECTION 11

Move Equipment

❖ Moves equipment objects from one location in the model to another. You can move equipment one piece at a time, or you can select multiple equipment objects and move them collectively.

The software maintains persistent relationships that exist between objects during a move. For example, if a pump has been mated to a surface, that relationship continues to exist even after the pump is moved to another location within the model. In some instances, relationships may prevent you from moving an object to where you want. For example, the pump cannot be moved off the surface, only to another location on the surface. You may need to delete relationships to move an object to its new location. Shapes, equipment objects, and ports added to an equipment object move as a rigid body when the equipment object is manipulated as a whole. To move an individual shape, select the shape and not the designed equipment object.

Move Ribbon

Specifies the **Move From** point and **Move To** point when you move an equipment object.

💡 **TIP** To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.

Move From

Defines the point of origin for moving the selected equipment object. This point serves as a fixed anchor during the move.

Move To

Defines the destination point for the selected equipment object.

Plane

Activates options for selecting a working plane for the move operation. The icon in the ribbon changes depending on your selection. The options include:

No Plane

Does not restrict movement to a plane.

Plan

Defines the work surface as the XY plane.

Elevation

Defines the work surface as the XZ plane.

Section

Defines the work surface as the YZ plane.

Name

Displays the name of the object selected.

System

Displays the parent system for the object selected.

Move Equipment with Precision Points

1. Select one or more equipment objects.
TIP To select multiple objects, hold CTRL as you click each object.
2. On the horizontal toolbar, click **Move** .
3. Click in a graphic view to define the **Move From** point. By default, this is the location that the equipment was in when you started the command.
4. Click in a graphic view to define the **Move To** point. For precise movements, use the options on the **PinPoint** ribbon.

NOTES

- If only one equipment object is selected, the point of origin defaults to the current position of the selected object.
- When you move an equipment object, the **Move To** point becomes the next **Move From** point.
- The software maintains relationships within the select set if they are still applicable after you have moved the objects.

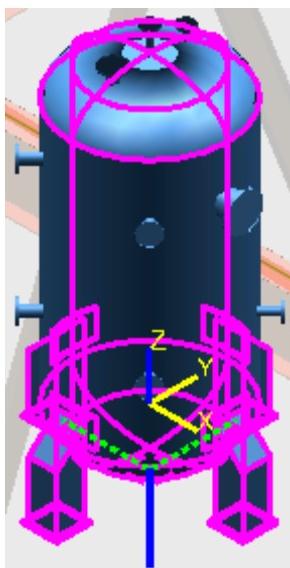
SECTION 12

Rotate Equipment

 Repositions equipment by rotating along a universal axis defined for the model view as well as an occurrence axis unique to the equipment itself. You can either select an object to rotate and then select the command, or select the command and then select the object to rotate. You can rotate equipment at any time.

By defining **Angle** and **Step** values, very precise rotations are possible. The axis of rotation is one of the axes of the current PinPoint coordinate system, one of the equipment's local coordinate system axes, a foundation port axis, or a nozzle axis. You can either type a value in the Angle box to define the rotation, or you can drag around the axis of rotation. This action causes the object to rotate dynamically about the axis, with a corresponding dynamic update of the Angle box. When rotating an object, you can also specify a Step value that represents the incremental angle value used by the software in rotating the piece of equipment.

When you select the **Rotate Equipment** command, the software displays the coordinate system of the selected equipment object and highlights the active axis, as shown in the accompanying graphic.



If you placed the equipment with a mate relationship to a surface, you can only rotate the equipment about the axis that is normal to the surface. When you start the command, the software selects this axis by default. If the equipment has more than one constraint, you cannot rotate it. The default point of rotation is the first foundation port of the equipment. If no foundation port exists, the origin becomes the default point of rotation.

Rotate Equipment Ribbon

Sets options for rotating equipment in the model. You cannot rotate equipment along an axis that is constrained by predefined positioning relationships, and you cannot rotate equipment that is fully constrained. For example, you cannot rotate a pump that is mated to a horizontal

reference plane and aligned with other pumps on either side, because in order to perform the rotation, one or more of these relationships is broken.

 **TIP** To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.

Rotation Point

Defines the origin point for the rotation of a piece of equipment. This point serves as a fixed anchor during the rotation.

Axes

Sets the geometrical axes to reference during the rotate operation. You can choose from the equipment's local coordinate system, the global coordinate system for the entire model, or any grid systems defined for the model.

Reference on Part

Instructs the system to use the selected part face or edge as the rotation reference, and selects the equipment's local coordinate system for the rotation of the part.

Reference on Model

Defines a reference in the model, and selects the global coordinate system for the rotation of the part.

Angle

Specifies the angle of rotation for the equipment. This option also dynamically displays the current angle during manual rotation.

Step

Defines an incremental value to use when rotating equipment. When you rotate equipment manually, the equipment only rotates to an angle equal to some multiple of the step value.

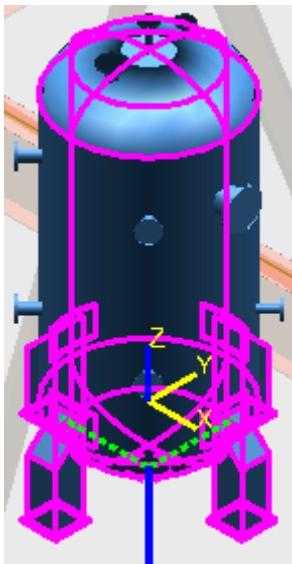
Close

Exits the command.

Rotate equipment

1. Select the piece of equipment to rotate.
2. Click **Rotate Equipment**  on the vertical toolbar.

The software displays the coordinate system of the selected equipment object and highlights the active axis, as shown in the accompanying graphic.



3. In the **Axes** box of the **Rotate Equipment** ribbon, select the axis, if needed. For more information on the ribbon, see Rotate Equipment Ribbon.
4. Type the incremental angle value needed in the **Step** box.
5. Specify the angle at which to rotate the equipment in the **Angle** box.
6. Click **Close** to exit the command.

NOTE The equipment object can also be rotated dynamically by dragging. When using this method, the **Angle** box updates dynamically as the equipment position is changed.

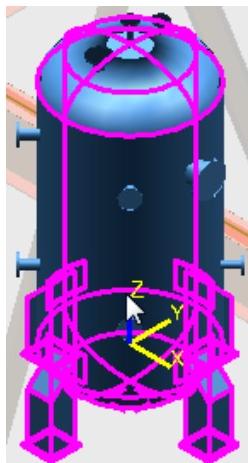
SECTION 13

Replace Equipment

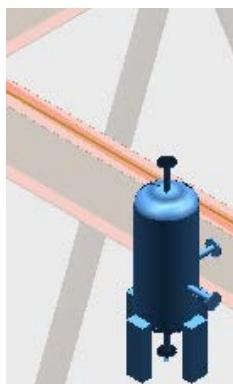
 Exchanges a selected equipment or equipment component in the model for a different item from the catalog. Only the origin and the local coordinate system of the original equipment object are retained.

Replace Equipment

1. Select the equipment or equipment component to replace.



2. Click **Replace Equipment**  on the vertical toolbar.
*The **Select Equipment** dialog box appears.*
3. Select the equipment to use in place of the selected item from the **Select Equipment** dialog box, and click **OK**.



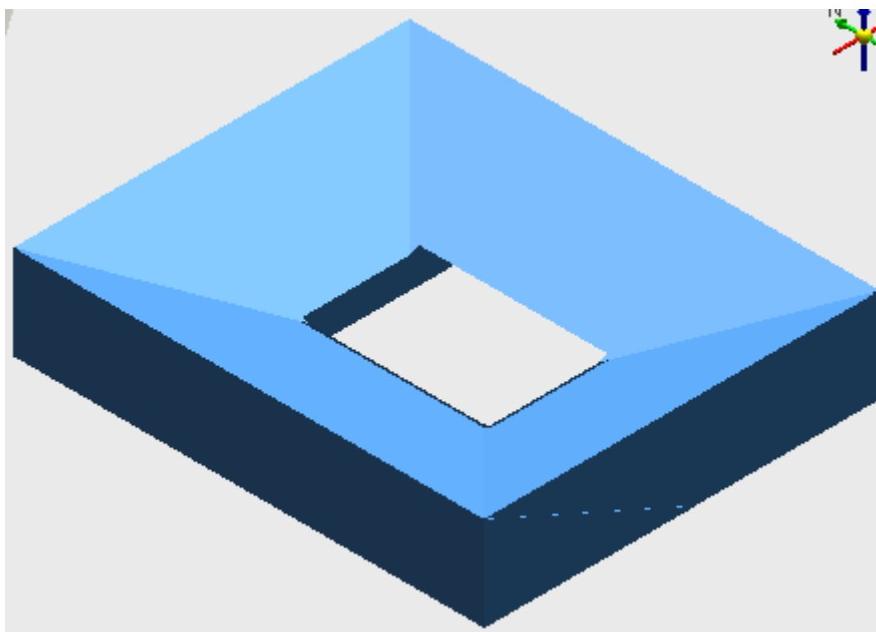
 **NOTE** Only the origin and the local coordinate system of the original equipment object are retained.

SECTION 14

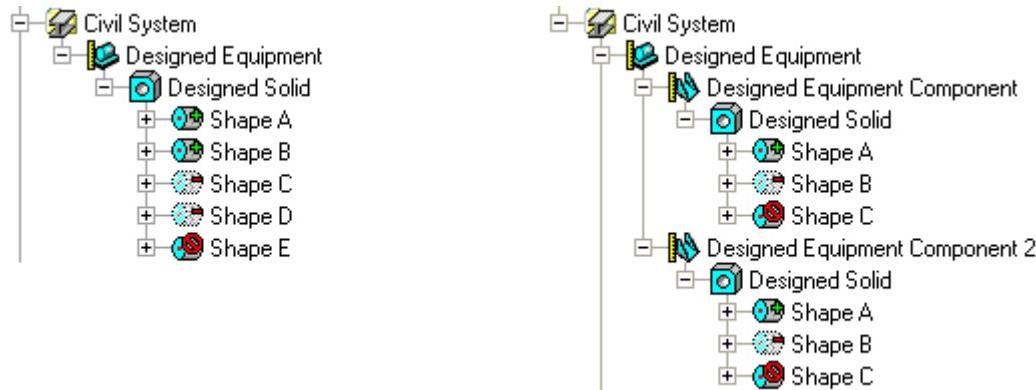
Place Designed Solid



The **Place Designed Solid** command is used to create highly customized designed solids for both complex concrete needs and equipment modeling.



Designed solids are essentially containers for a collection of shapes that you add to and subtract from the solid. To place a designed solid, a designed equipment or a designed equipment component must first be placed in the model. Then, the designed solid is placed as a child of the designed equipment or the designed equipment component. You can then add shapes underneath the designed solid. You can have multiple designed solids under one designed equipment or designed equipment component. You can also have multiple designed equipment components under one designed equipment.

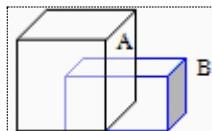


A shape must be a child of a designed solid in order for the software to compute the weight and center-of-gravity. The designed solid's weight and center-of-gravity properties are the sum of all shapes underneath a designed solid. For example, Shape A + Shape B + Shape C = Total Surface Area, Volume, Weight and Center-of-Gravity of the designed solid. If there is more than one solid underneath a designed equipment parent, these solids are also summed up to give a Total Weight and Center-of-Gravity for the designed equipment. For example, Solid A + Solid B + Solid C = Total Weight and Center-of-Gravity of the designed equipment.

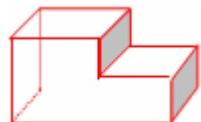
★IMPORTANT Only shapes with simple physical or detailed physical aspects are included in weight and center-of-gravity calculations for designed solids. In addition, if some child shapes are displayed using the simple physical aspect and other child shapes of the same designed solid are displayed using the detailed physical aspect, only those shapes displayed as detailed physical are included in the weight and center-of-gravity calculations. If all the child shapes of a designed solid are displayed using the simple physical aspect, then they are all included in weight and center-of-gravity calculations. We recommend that you do not mix simple physical and detailed physical aspects in the same designed solid, as doing so will cause incorrect weight and center-of-gravity values.

Each shape that you add to the designed solid can add material to the parent solid, remove material from the parent solid, or not affect the parent (suppressed). The shape icon shown in Workspace Explorer represents an **Add** , **Subtract** , or **Suppress**  operation. By default, shapes are added to a solid; however, during placement or after placement, you can change the shape operation to **Subtract Shape** or **Suppress Shape**.

Think of **Add Shape** and **Subtract Shape** as Boolean operations. For example, the designed solid below contains two shapes, A and B:



If you add shape B to shape A, the resulting designed solid appears as follows:



However, if you subtract shape B from shape A, the resulting designed solid appears as follows:



The Suppress Shape operation is used for creating construction geometries for reference when placing other shapes. The suppressed shape itself is not included in the designed solid; therefore, it will not affect weight and center-of-gravity calculations.

Place Designed Solid Ribbon

Sets options for adding designed solids to your model. This ribbon displays automatically after you select the **Place Designed Solids**  command, or when you select an existing designed solid.

Designed Solid Properties

Edits the occurrence properties and reviews the static properties of an existing designed solid. Properties can be edited only after the object is placed in the model. For more information, see *Solid Properties Dialog Box* (on page 141).

Relationship List

Lists all relationships for the selected designed solid and provides an option for creating a new relationship if the designed solid is not already fully constrained. A designed solid part is fully constrained when it has sufficient defined relationships to prevent movement or rotation of the part along all three coordinate axes.

Positioning Relationships

Displays the available options for types of positioning relationships. Some options may not be available for all designed solid types. See *Positioning Relationships* (on page 12) for more information.

Delete Relationship

Removes the selected relationship from the designed solid in the model and in the database. Using the **Relationships** list box, select a previously-existing relationship for the designed solid, and click **Delete Relationship**. You can use this command only when modifying existing designed solids.

Shape Reference

Prompts you for the reference on the designed solid that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

Second Part Reference

Prompts you for the reference on the designed solid or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first shape part chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial designed solid placement, if needed.

Name

Displays the designed solid name, as dictated by your predefined name rules, and accepts changes to that name.

Equipment

Displays the name of the parent designed solid. This field only displays when you select a shape that comprises the designed solid.

Operators List

Activates the **Order Shapes** dialog box, with which you can control the order in which the software processes the shapes that comprise the designed solid. Controlling the order in which the shapes are processed can be important when some shapes that remove material from the designed solid overlap with other shapes that add material to the designed solid.

Order Shapes Dialog Box

Controls the order in which the designed solid's shapes are processed by the software, which can be very important when a shape that cuts material from the designed solid overlaps a shape that adds material to the designed solid. The designed solid could look very different depending on which shape, the cut or the add, the software processes last.

This dialog box is activated by **Operators List**  on the Modify Designed Solid ribbon.

Shape Name

Displays the name of the shape. You can select the shape name in the list to highlight that shape in the model view.

Operator Type

Displays whether the shape adds material to the designed solid or subtracts material from the designed solid.

Up

Moves the selected shape up one row in the list.

Down

Moves the selected shape down one row in the list.

Top

Moves the selected shape to be the first shape in the list.

Bottom

Moves the selected shape to the last shape in the list.

OK

Applies the changes made to the list to the designed solid and closes the dialog box.

Cancel

Ignores any changes made to the list and closes the dialog box.

Apply

Applies the changes made to the list to the designed solid, but leaves the dialog box active. Use this option to see changes that you make instantly.

What do you want to do?

- *Create a basic solid* (on page 139)
 - *Import shape to designed solid* (on page 140)
 - *Switch add, subtract, or suppressed shape* (on page 141)
 - *Order shapes in designed solid* (on page 141)
-

Create a basic solid

1. Place a designed equipment or a designed equipment component using the **Equipment > Civil > Miscellaneous** equipment type.

Place designed equipment (on page 41)

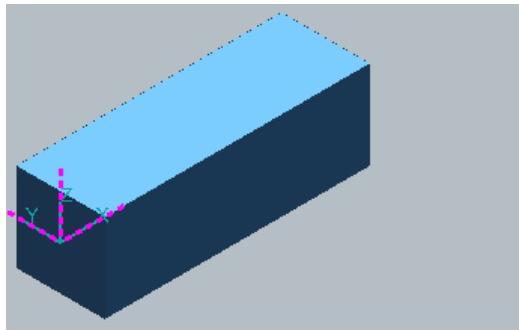
Place designed equipment component (on page 69)

TIP You must use the Equipment > Civil > Miscellaneous equipment type for the software to correctly calculate the mass properties of the solid (volume, surface area, weight, and center of gravity).

2. With the designed equipment or designed equipment component that you created in step 1 selected, click **Place Designed Solid** .

The software adds the new designed solid as a child of the designed equipment or equipment component.

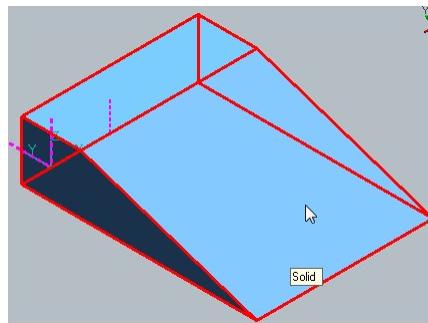
3. Click **Place Shape** , and then select the first shape for your designed solid.
4. In the **Shape Properties** dialog box, type the dimensions of your first shape, and then click **OK**.



The software adds the shape to the designed solid in Workspace Explorer and places the shape in the model at the point that you indicate.

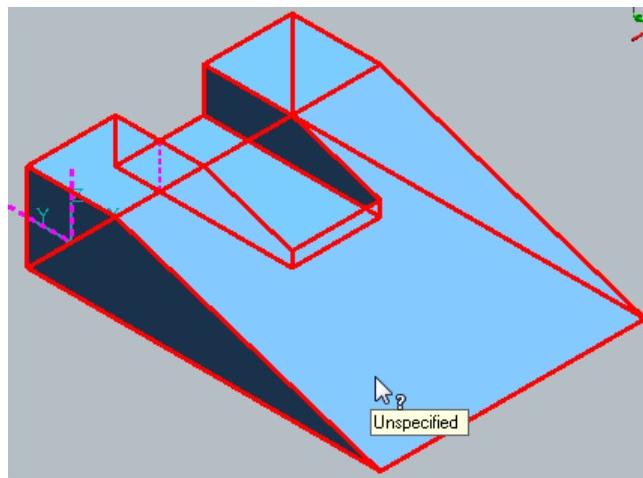
5. On the ribbon, select whether the shape adds , subtracts , or is suppressed . Use suppressed for construction graphics and for shapes you want to temporarily remove from affecting the design solid.
6. Click **Place Shape** , and then select the next shape for your designed solid.
7. In the **Shape Properties** dialog box, type the dimensions for the second shape, and then click **OK**.

- Identify the location of the second shape in the model.



TIP If you are placing a cylinder, cone, or eccentric cone shape, use the **Shape Reference** option on the toolbar to select the end plane of the cylinder or cone when using the Mate relationship to a plane of another shape.

- On the ribbon, select whether the second shape adds +, subtracts -, or is suppressed .
- Continue to added shapes to the designed solid using the add, subtract, and suppress options as needed.



Import shape to designed solid

- Click **Place Imported Shape from File** on the vertical toolbar.
- Select the parent designed solid in a graphic view or in the **Workspace Explorer**.
- In the **Select Shape File** dialog box, navigate to and select the SAT file or MicroStation DGN file that contains the shape to insert. Click **OK**.
- In the **Display Aspect** list box, select the aspects from the list.
TIP You can click **Format > View** and change the **Render Selected Aspects** option in the **Format View** dialog box to display the aspect in the active graphic view.
- Click **OK**.

6. Click in the graphic view to select an approximate location or reference element for a relationship for the shape.

SAT File Limitations

- From a performance point of view, the SAT file size should be less than 5 MB.
- From a drawings point of view, the files must not be detailed and must not spread across a large space as they are caught in volume filters for Drawings.
- Spreading the SAT files across multiple equipment will reduce the volume of each .sat file.
- For efficient Interference Checker processing, SAT files should not have a footprint (range) greater than 100 meters.

Switch add, subtract, or suppressed shape

1. Click **Select**  on the vertical toolbar.
2. Set the locate filter to **Shape**.
3. In a graphic view or in Workspace Explorer, select the shape to edit.
4. On the ribbon, select whether the shape adds , subtracts , or is suppressed .

Order shapes in designed solid

1. Click **Select**  on the vertical toolbar.
 2. Set the locate filter to **Solids**.
 3. Select the parent designed solid whose shapes you want to re-order.
 4. Click **Operators List**  on the ribbon.
 5. In the **Order Shapes** dialog box, use the **Up**, **Down**, **Top**, and **Bottom** commands to order the shapes as needed.
 6. Click **Apply** or **OK** to save your changes and update the designed solid.
- TIP** See *Place Designed Solid* (on page 135) for more information on why the order of shapes in a solid is important.

Solid Properties Dialog Box

Displays designed solid properties for review and editing.

See Also

Occurrence Tab (Solid Properties Dialog Box) (on page 142)

Configuration Tab (on page 148)

Notes Tab (on page 150)

Occurrence Tab (Solid Properties Dialog Box)

Displays all editable instance-specific information about the selected designed solid. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If more than one solid object is selected, common occurrence properties for the selected objects appear on the tab.

When viewing properties for a single solid object, the following properties appear. More properties may appear depending on what you defined in the reference data. For more information about occurrences defined in the reference data, see the *Equipment and Furnishings Reference Data Guide* available from the **Help > Printable Guides** command in the software.

NOTE Because equipment properties are customizable in the equipment and furnishings reference data, only the properties that are required by the software are documented.

Category

Select the properties to view, modify, or define. Equipment properties are divided into several different categories: Standard, Weight and CG, and Position and Orientation.

Display Aspect

Select the display aspects that you want to see for the solid.

Standard

Name

Displays the name of the solid object. The solid name is based on the **Name Rule** selection. If you type a name in this field, the **Name Rule** property updates to **User Defined**.

Name Rule

Specify the naming rule to use to name this solid object. You can select one of the listed rules or select **User Defined** to specify the solid name yourself in the **Name** box.

Material Name

Select the material for the solid.

Material Grade

Select the material grade for the solid.

NOTE You must define a material and material grade before the software can calculate the solid's weight and center of gravity.

Surface Area

Displays the calculated surface area.

Volume

Displays the calculated volume.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material

density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin

Specifies the way in which the wet weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Position and Orientation

East

Displays the distance of the connection point from the active coordinate system origin in the east direction.

North

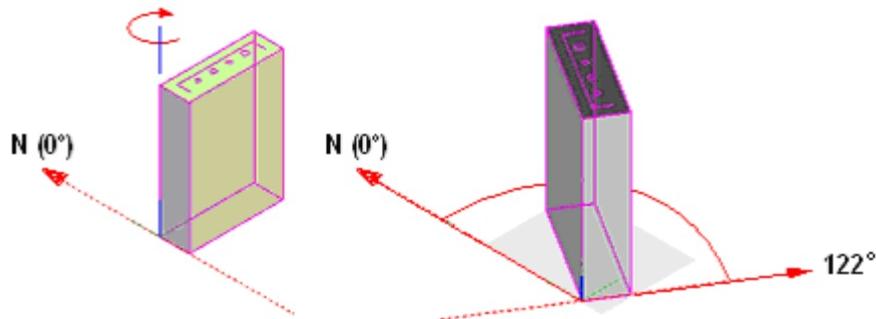
Displays the distance of the connection point from the active coordinate system origin in the north direction.

Elevation

Displays the distance of the connection point above or below the active coordinate system origin.

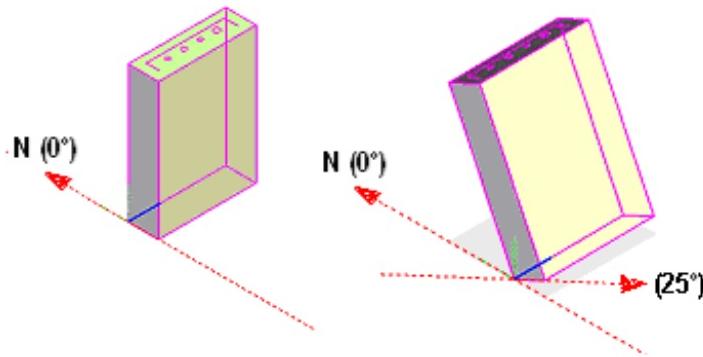
Bearing

Specifies the bearing angle for the object. The **Bearing** angle is measured between the local x-axis of the object and the Y-axis (North) of the global coordinate system in the XY-plane. The local x-axis is the default axis of primary symmetry for all symbols in the catalog. The bearing measurement direction is clockwise from the active coordinate system North looking in the negative active coordinate system direction; that is, down from 0 to 360 degrees. You can enter negative bearing angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the bearing measure displays 0 degrees.



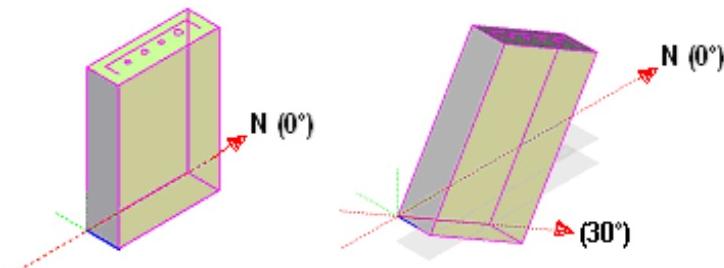
Pitch

Specifies the pitch angle for the shape. The **Pitch** angle measures between the x-axis of the object and the X-axis of the global coordinate system in the XZ-plane. This option sets the reference in the model to a line that is the intersection of the vertical plane through the X-axis of the local coordinate system and the active coordinate system horizontal plane. The angle is measured in the positive direction from the horizontal plane in the active coordinate system up direction regardless of the current bearing. Another way to describe pitch is the rotation of the object about its y-axis. Pitch angles are limited to between -90 degrees and +90 degrees, with 0 indicating horizontal.



Roll

Specifies the roll angle for the object. The **Roll** angle measures between the local z-axis of the object and the Z-axis of the global coordinate in the YZ-plane. Another way to describe roll is the rotation of the object about its x-axis. This option sets the reference in the model to a line that is perpendicular to the local coordinate system x-axis and in the horizontal plane. Roll angles are measured clockwise from horizontal to the y-axis of the local coordinate system. The roll angle is between 0 and 360 degrees. You can enter negative roll angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the reference in the model is the North axis.



SECTION 15

Editing Properties and Relationships

After you have placed an equipment object in the model, it might be necessary to adjust its properties or its relationships with other objects in the model. The Equipment and Furnishings task provides you with the flexibility to make design changes at any point in the design process.

All items that compose an equipment object have properties that you can edit. Using the **Select** command on the vertical toolbar, you select the object to edit. An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon.



The **Locate Filter** box contains the available, pre-defined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered items in a graphic view and in the **Workspace Explorer**. For example, if you select **Shape**, you can select only shapes that have been placed in a graphic view or in the **Workspace Explorer**.

The Equipment and Furnishings task includes these filters:

Cable Nozzle

Limits the selection of items to cable nozzles within an equipment object.

Cable Tray Nozzle

Limits the selection of items to cable tray nozzles within an equipment object.

Conduit Nozzle

Limits the selection of items to conduit nozzles within an equipment object.

Construction Graphics

Limits the selection of items to construction graphics.

Control Points

Limits the selection of items to the control points of an equipment object.

Equipment

Limits the selection of items to only equipment items that have been placed from the catalog or as designed equipment items.

Equipment Component

Limits the selection of items to a component of the equipment object.

Foundation Port

Limits the selection of items to the foundation port of the equipment object.

HVAC Nozzle

Limits the selection of items to HVAC nozzles within an equipment object.

Pipe Nozzle

Limits the selection of items to pipe nozzles within an equipment object.

Shape

Limits the selection of items to the individual shapes that compose a designed equipment object in a graphic view or in the **Workspace Explorer**.

Solids

Limits the selection of items to solids within an equipment object.

All

Allows the selection of any object, even objects created in another task.

More

Displays the **Select Filter** dialog box. Displays the **Select Filter** dialog box. The **Select Filter** dialog box displays all of the simple filters that are set up to accept object type or system objects as discrimination data. You can then select additional filters to add them to this list. Only the selection of filters is available; all other options are disabled.

**Inside Fence**

Selects all objects entirely inside the fence.

**Inside/Overlap Fence**

Selects all objects entirely inside the fence and those outside but touching the fence at some point.

The **Properties** dialog box allows you to adjust any of the occurrence properties of a particular piece of equipment, shape, or nozzle; not only during initial placement, but also afterward, including properties that may change the size or shape of the equipment, shape, or nozzle.

★IMPORTANT If the equipment model was designed using Solid Edge, modifying any occurrence properties also requires that Solid Edge be installed on your computer.

The **Positioning Relationship** box sets necessary constraints for a piece of equipment or shape either at placement or afterward. Because shapes, equipment objects, and ports added to an equipment object move as a rigid body when the equipment object is manipulated, you must avoid creating positioning relationships that conflict with this rigid body behavior when adding objects to an equipment object. With this in mind, relationships should be restricted to only those objects belonging to the designed equipment object.

See Also

- [Edit Equipment Relationships \(on page 27\)](#)*
- [Edit Equipment Properties \(on page 26\)](#)*
- [Edit Designed Equipment Properties \(on page 42\)](#)*
- [Edit Prismatic Shape Properties \(on page 76\)](#)*
- [Edit Shape Properties \(on page 75\)](#)*

Common Property Tabs

The software displays some common property tabs on the properties dialog boxes for all equipment objects. Instead of repeatedly listing the common tabs with each equipment object property dialog box, they are documented here for easy reference.

See Also

- [General Tab \(Reference 3D Properties\) \(on page 149\)](#)
- [Configuration Tab \(on page 148\)](#)
- [Connection Tab \(on page 37\)](#)
- [Notes Tab \(on page 150\)](#)

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

General Tab (Reference 3D Properties)

The **General** tab displays the properties that are automatically determined by the software at placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you selected more than one reference 3D object, and then selected the **Properties** command, only the common properties between the selected objects display.

Category

Select the properties that you want to view. Properties are grouped into several different categories. Use the menu list to see the available categories for your selected reference 3D object.

Connection Tab

Displays information on the connection points of a piece of equipment, including piping, electrical, foundation, or HVAC connections, and Reference 3D objects. If more than one equipment object is selected, only the common properties of the connections for the selected objects display on the tab. For more information about the information defined in the reference data, see the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Connector

Select the connector for which to view properties. With the exception of Reference 3D object connectors, the connectors display in alphabetical order. When you select a connector from the list, the graphic object associated with the connection, if one exists, highlights in the graphic view for further visual confirmation of the port selected.

Property

Lists all the properties of the selected connection. These properties vary depending on the type of connection selected.

Value

Displays the value of the corresponding property.

Definition Tab (Equipment Properties Dialog Box)

Displays the part information for the equipment object, the properties and their values, as defined in the reference data. If more than one equipment object is selected, only the common properties for the selected objects display on the tab. For more information about the information defined in the reference data, see the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Category

Select the properties to review. Equipment information is divided into several different categories, such as **Standard**, **Equipment Specification**, **Equipment Support**, **Nozzle Length**, and **Equipment Dimensions**.

★ IMPORTANT Not all categories are available for all equipment objects.

Property

Displays the name of the property. The properties that appear are dependent on the equipment type selected. For example, the properties displayed for an equipment object based on a pump are different from those based on a cooler. For more information on equipment part properties, see the *Equipment and Furnishings Reference Data Guide*, available with the **Help > Printable Guides** command within the Equipment and Furnishings task.

Value

Displays the value of the corresponding property.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

NOTE Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

SECTION 16

Correlate Existing Equipment for Use in an Integrated Environment

If the nozzles exist on the equipment before you correlate, then the software attempts to correlate the nozzles automatically. Otherwise, you must correlate existing equipment in two processes. First, correlate the body. Then, correlate the nozzles in a separate process.

Correlate the Body

1. Select **Correlate with Design Basis**.
2. Select the main equipment body.
3. Select the equipment on the P&ID.

Correlate the Nozzles

1. Select **Correlate with Design Basis**.
2. Select a nozzle on the equipment.
3. Select the corresponding nozzle on the P&ID.

NOTE If the nozzle names in the 3D model and the names identified in the P&ID match, then the nozzles correlate to the correct state. If there are any discrepancies, you can view them using the **SmartPlant > Compare Design Basis** command.

Correlate New Equipment for Use in an Integrated Environment

1. Select the equipment from the P&ID.
2. Select **Place Equipment**  to create the equipment folder, and select the matching equipment by type using the catalog.
3. Place the shape, and select the equipment parent.
4. Select the nozzle from the P&ID.
5. Click **Place Nozzle** .
6. Select the designed equipment parent for the nozzle.
7. Position the nozzle on the equipment.

NOTE You must type all of the size, end preparation, termination class, and rating practice information for the first nozzle that you place. The software reads all subsequent nozzle information directly from the P&ID.

Update Equipment for Use in an Integrated Environment

1. Click **Compare Design Basis**.
2. Select the equipment folder from the **Workspace Explorer**, or select the symbol from the P&ID.
3. Click **Update**.

NOTES

- Updating adjusts all of the nozzles associated with the main body as well as the equipment body itself. The nozzles and the equipment appear highlighted in the **Workspace Explorer**. If they are not highlighted then you can select the equipment in the **Workspace Explorer** and choose **Select Nested**. Then, select **SmartPlant > Compare Design Basis**. Select **Update** to update the equipment and all associated nozzles.
- If you change dimensions or names of a nozzle, then you must update that nozzle separately.

Glossary

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

arrangement (machinery)

Machinery arranged in three-dimensional space with accurate dimensional representation for installation.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

basic design

Engineering definition of the model and its systems.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog database

The database that contains the reference data. Each model database can reference a different Catalog database.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

class

Grouping of individual objects that share some very significant, common characteristics.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

component

Physical part that a feature generates.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

cutting plane

A plane that cuts through an object.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes after the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

degree

The highest polynomial factor in the curve or surface mathematical definition. A line is a degree 1 curve, while a cubic B-spline is a degree 3 curve.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so you can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

detail schedule

Lowest level of schedule used to manage and track work progress.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

easting

A term that describes an east coordinate location in a coordinate system.

edge

A topological object that represents a trimmed curve bounded by a start and end vertex.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

equipment modeler

Facility of the software to create three-dimensional representations of equipment and components for use in defining arrangements.

fabricate

To cut, punch, and sub-assemble members in the shop.

face-to-face

The overall length of a component from the inlet face to the outlet face.

fasteners

Bolts and rivets used to connect structural members.

element

Primitive geometric shape such as a line, circle, or arc.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

focus of rotation

A point or line about which an object or view turns.

full penetration weld

A type of weld in which the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts such as movable articles and fittings that normally are not associated with a system (for example, a chair).

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been

loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

host location

The first location created for a Site. This host location is defined when the Database Wizard creates the Site database.

host server

The database server on which the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

initial design

Early stage of design work, generally before contract, used to estimate construction costs and provide a rough concept of the intended plant. Contains information relating to a plant created during its initial (concept) design period.

initial structural plan

Principal structural plan for the plant; also called a construction profile.

instantiation

Occurrence of a catalog object at a specific geometric location in the model.

interference checking

A process that identifies possible collisions or insufficient clearance between objects in the model.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

kinematics analysis

Analysis of mechanical motion.

ksi

Kips per square inch.

leg length analysis

Preferred term is welding length analysis.

library

Resource of reference information that you can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load group

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the server where the Site databases reside for that Location. A Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

node

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

object

A type of data other than the native graphic format of the application.

occurrence (of part or equipment)

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa:' in the reference data workbooks. You can view and modify occurrence

properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

parameter

A property whose value determines the characteristics or behavior of something.

part class

A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part number

Unique identifier of a part.

PDS (Plant Design System)

A comprehensive, intelligent, computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

query select sets

Set of objects that are selected in a query or queries on the database.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after you modify or add to the database structure.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

specifications

Contracted requirements for the plant.

stud

A bolt, threaded on both ends, used to connect components.

suspended floor

A concrete floor system built above and off the ground.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

tolerant geometry

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

trimmed surface

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

vertex

A topological object that represents a point in the three-dimensional model.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

wirebody

A topological object that represents a collection of edges jointed at their common endpoints.

wizard

Software routine attached to an application that provides guidance and expert help to you to complete one of the functionalities of the application.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

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